HEIDENHAIN

TNC 426

NC-Software
280 462 xx
280 463 xx

User's Manual
Conversational
Programming

4/97

Controls on the visual display unit

⊙ ▭ ▭   Toggle display between machining and programming modes

⊙ GRAPHICS TEXT SPLIT SCREEN   Split screen layout

▢   Soft keys for selecting functions in screen

◁ ▷   Shift soft-key rows for the soft keys

☼ ◑   Brightness, contrast

Typewriter keyboard for entering letters and symbols

Q W E R T Y   File name Comments

G F S T M   ISO programs

Machine operating modes

🖑   MANUAL OPERATION

⊛   ELECTRONIC HANDWHEEL

▤   POSITIONING WITH MDI

▣   PROGRAM RUN, SINGLE BLOCK

⇥   PROGRAM RUN, FULL SEQUENCE

Programming modes

◈   PROGRAMMING AND EDITING

⇥   TEST RUN

Program/file management, TNC functions

PGM MGT   Select or delete programs and files External data transfer

PGM CALL   Enter program call in a program

MOD   MOD functions

HELP   HELP functions

CALC   Pocket calculator

Moving the cursor, going directly to blocks, cycles and parameter functions

↑ ↓ ← →   Move highlight

GOTO ▢   Go directly to blocks, cycles and parameter functions

Override control knobs for feed rate/spindle speed

Programming path movements

APPR DEP   Approach/depart contour

L↗   Straight line

CC   Circle center/pole for polar coordinates

↻ C   Circle with center

CR   Circle with radius

CT↗   Tangential circle

CHF   Chamfer

RND   Corner rounding

Tool functions

TOOL DEF   TOOL CALL   Enter or call tool length and radius

Cycles, subprograms and program section repeats

CYCL DEF   CYCL CALL   Define and call cycles

LBL SET   LBL CALL   Enter and call labels for subprogramming and program section repeats

STOP   Program stop in a program

TOUCH PROBE   Enter touch probe functions in a program

Coordinate axes and numbers, editing

X … V   Select coordinate axes or enter them in a program

0 … 9   Numbers

•   Decimal point

⁻/+   Change arithmetic sign

P   Polar coordinates

I   Incremental dimensions

Q   Q parameters

✛   Capture actual position

NO ENT   Skip dialog questions, delete words

ENT   Confirm entry and resume dialog

END ▢   End block

CE   Clear numerical entry or TNC error message

DEL ▢   Abort dialog, delete program section

TNC Models, Software and Features

This manual describes functions and features provided by the TNCs with the following NC software numbers.

TNC Model	NC Software No.
TNC 426 CA, TNC 426 PA	280 462 xx
TNC 426 CE, TNC 426 PE	280 463 xx

The suffix E indicates the export versions of the TNC, which have the following limitations:

■ Input and machining accuracy are limited to 1 μm.

■ Linear movement is possible in no more than 4 axes simultaneously

The machine tool builder adapts the useable features of the TNC to his machine by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by your machine tool.

TNC functions that may not be available on your machine include:

■ Probing function for the 3-D touch probe

■ Digitizing option

■ Tool measurement with the TT 120

■ Rigid tapping

■ Returning to the contour after an interruption

Please contact your machine tool builder to become familiar with the individual implementation of the control on your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

Location of use
The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Contents

Contents

Contents

Contents

8 PROGRAMMING: CYCLES 129

Contents

1

Introduction

1.1 The TNC 426

HEIDENHAIN TNC controls are shop-floor programmable contouring controls for milling, drilling and boring machines, and machine centers with up to five axes.

You can program conventional milling, drilling and boring operations right at the machine with the easily understandable interactive conversational guidance. You can also change the angular position of the spindle under program control.

An integrated hard disk provides storage for as many programs as you like, even if they were created off-line or by digitizing. For quick calculations you can call up the on-screen pocket calculator at any time.

Keyboard and screen layout are clearly arranged in a such way that the functions are fast and easy to use.

Programming: HEIDENHAIN conversational and ISO formats
HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming carries out the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining. It is also possible to program in ISO format or DNC mode.

You can also enter and test one program while the TNC is running another.

Compatibility
The TNC can execute all part programs that were written on HEIDENHAIN controls TNC 150 B and later.

1.2 Visual Display Unit and Keyboard

Visual display unit

The figure at right shows the keys and controls on the VDU:

1 Setting control for brightness and contrast

2 Shift key for switchover between machining and programming modes

3 Setting the screen layout

4 Soft key selector keys

5 Switching the soft-key rows

6 Header
When the TNC is on, the selected operating modes are written in the screen header: the machining mode to the left and the programming mode at right. The currently active mode is displayed in the larger box, where the dialog prompts and TNC messages also appear.

7 Soft keys
In the bottom line, the TNC indicates additional functions in a soft-key row. You can select these functions with the keys 4 located below. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The line representing the active soft-key row is highlighted.

Screen layout

You select the screen layout yourself: In the PROGRAMMING AND EDITING mode of operation, for example, you can have the TNC show program blocks in the left screen window while the right window displays programming graphics. You could also display the program structure in the right window instead, or only display program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:

Press the SPLIT SCREEN key: The soft-key row shows the available layout options.

```
  PGM
   +
GRAPHICS
```
Select the desired screen layout.

Keyboard

The figure at right shows the keys of the keyboard grouped according to their functions:

1 Alphanumeric keyboard
for entering texts and file names, as well as for programming in ISO format

2 File management,
pocket calculator,
MOD functions,
HELP functions

3 Programming modes

4 Machine operating modes

5 Initiation of programming dialog

6 Arrow keys and GOTO jump command

7 Numerical input and axis selection

The functions of the individual keys are described in the foldout of the front cover. Machine panel buttons, e.g. NC START, are described in the manual for your machine tool.
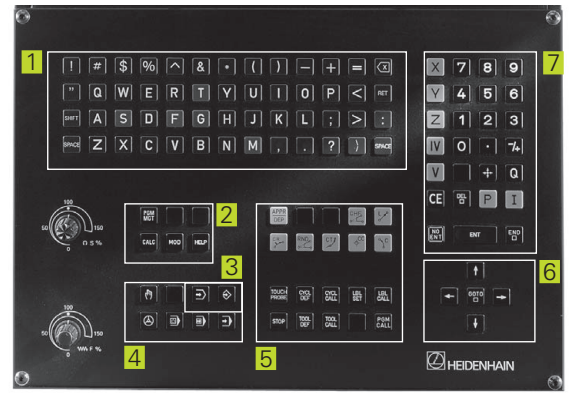
1.3 Modes of Operation

The TNC offers the following modes of operation for the various functions and working steps that you need to machine a workpiece:

MANUAL OPERATION and ELECTRONIC HANDWHEEL

The MANUAL OPERATION mode is required for setting up the machine tool. In this operating mode, you can position the machine axes manually or by increments, set the datums, and tilt the working plane.

The ELECTRONIC HANDWHEEL mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Soft keys for selecting the screen layout
(select as described previously)

Soft key	Screen windows
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display

POSITIONING WITH MANUAL DATA INPUT (MDI)

This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning. You can also define point tables for setting the digitizing range in this mode.

Soft keys for selecting the screen layout

Soft key	Screen windows
PGM	Program blocks
PGM + STATUS	Left: program blocks, right: status display

```
POSITIONING WITH MANL.DATA INPUT    PROGRAMMING
                                    AND EDITING

0  BEGIN PGM $MDI MM
1  TOOL CALL 1 Z S4500                NOML.
2  L Z+100 R0 F MAX                    X  +250.0000
3  L X-20 Y+5 R0 F MAX M3              Y  +102.3880
4  L Z-5 R0 F2000                      Z  -114.0914
5  L X+120 F600                        C   +30.0000
6  L IV+30 F2000                       B   +90.0000
7  L X-20 F600
8  END PGM $MDI MM                                   B +90.0000
                                                     C +30.0000

                                        BASIC ROTATION   +0.0000

ACTL.   +X  +250,0000    +Y  +102,3880
        +Z  -114,0914    +C   +30,0000
        +B   +90,0000
T                        F 0           M 5/9
```

| STATUS PGM | STATUS POS. | STATUS TOOL | STATUS COORD. TRANSF. | STATUS TOOL PROBE | | PNT | |

PROGRAMMING AND EDITING

In this mode of operation you can write your part programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps, or you can use a separate screen window to prepare your program structure.

Soft keys for selecting the screen layout

Soft key	Screen windows
PGM	Program blocks
PGM + SECTS	Left: program blocks, right: program structure
PGM + GRAPHICS	Left: program blocks, right: programming graphics

```
MANUAL        PROGRAMMING AND EDITING
OPERATION

42 CP PA+0 DR- R0 FQ6           - PARAMETER DEFINITION
43 L Z+08 R0 F9998              - MAKE POCKET
44 FN 1: Q20 = +Q20 + +8.8        - ROUGH OUT
45 FN 9: IF +1 EQU +1 GOTO LBL 111   - FINISHING
46 * - MAKE HOLE PATTERN        - MAKE HOLE PATTERN
47 *  - CENTER DRILL              - CENTER DRILL
48 LBL 47                         - PECKING
49 L X+0 Y+1 R0 F9998             - TAPPING
50 L Z+08                       END PGM FGLIEDER
51 L Z+09 R0 FQ7
52 L X+5 Y+10 R0 FQ6
53 L Z+08 R0 F9998
54 FN 1: Q20 = +Q20 + +8.8
55 FN 9: IF +1 EQU +1 GOTO LBL 111
56 *  - PECKING
```

| FL | FLT | FC | FCT | FPOL | BLK FORM | INSERT SECTION | CHANGE WINDOW |

TEST RUN

In the TEST RUN mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the work space. This simulation is supported graphically in different display modes.

Soft keys for selecting the screen layout
Same as in the PROGRAM RUN operating modes on the next page.

```
MANUAL        TEST RUN
OPERATION

37 CVCL DEF 6.2 PECKG 4 F100 ALLOW +0
38 CVCL DEF 6.3 ANGLE +0 F800
39 L Z+2 R0 F9999 M99
40 CVCL DEF 14.0 CONTOUR GEOMETRY
41 CVCL DEF 14.1 CONTOUR LABEL 1 /2
   /3 /4 /5 /7
42 CVCL DEF 14.2 CONTOUR LABEL 8 /9
   /10 /11 /12
43 CVCL DEF 6.0 ROUGH-OUT
44 CVCL DEF 6.1 SET UP 18 DEPTH -8
45 CVCL DEF 6.2 PECKG 4 F300
   ALLOW +0.7
46 CVCL DEF 6.3 ANGLE +0 F600
47 CVCL CALL                        X +70.4  Y +55.5
48 CVCL DEF 14.0 CONTOUR GEOMETRY          64:19:13
```

| | | | OFF/ON | START SINGLE | STOP AT N | START | RESET + START |

PROGRAM RUN, FULL SEQUENCE and PROGRAM RUN, SINGLE BLOCK

In the PROGRAM RUN, FULL SEQUENCE mode of operation the TNC executes a part program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the PROGRAM RUN, SINGLE BLOCK mode of operation you execute each block separately by pressing the machine START button.

Soft keys for selecting the screen layout

Soft key	Screen windows
PGM	Program blocks
PGM + SECTS	Left: program blocks, right: program structure
PGM + STATUS	Left: program blocks, right: STATUS
PGM + GRAPHICS	Left: program blocks, right: graphics
GRAPHICS	Graphics

```
PROGRAM RUN, FULL SEQUENCE        PROGRAM TABLE
                                  EDITING

0  BEGIN PGM FK1 MM
1  BLK FORM 0.1 Z X+0 Y+0 Z-20
2  BLK FORM 0.2 X+100 Y+100 Z+0
3  TOOL CALL 1 Z S500
4  L Z+250 R0 F MAX
5  L X-20 Y+30 R0 F MAX
6  L Z-10 R0 F1000 M3
7  APPR CT X+2 Y+30 CCA90 R+5 RL F250
8  FC DR- R18 CLSD+ CCX+20 CCY+30

                                  0°         00:00:00
ACTL.     +X    +250,0000    +Y   +102,3880
          +Z    -114,0914    +C    +30,0000
          +B     +90,0000
T                           F  0          M 5/9
```

1.4 Status Displays

"General" status displays

The status display informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

- PROGRAM RUN, SINGLE BLOCK and PROGRAM RUN, FULL SEQUENCE, except if the screen layout is set to display graphics only, and

- POSITIONING WITH MDI

In the operating modes MANUAL OPERATION and ELECTRONIC HANDWHEEL the status display appears in the large window.

Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
X Y Z	Machine axes
S F M	Spindle speed S, feed rate F and active M functions
✳	Program run started
→|←	Axis locked
⊕	Axis can be moved with the handwheel
◹	Axes are moving in a tilted working plane
◹	Axes are moving under a basic rotation

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes, except in the PROGRAMMING AND EDITING mode of operation.

To switch on the additional status display:

Call the soft-key row for screen layout.

PGM + STATUS   Select the layout option for the additional status display.

```
PROGRAM RUN, SINGLE BLOCK          PROGRAM TABLE
                                   EDITING

0  BEGIN PGM FK1 MM
1  BLK FORM 0.1 Z X+0 Y+0 Z-20
2  BLK FORM 0.2 X+100 Y+100 Z+0
3  TOOL CALL 1 Z S500
4  L Z+250 R0 F MAX
5  L X-20 Y+30 R0 F MAX
6  L Z-10 R0 F1000 M3
7  APPR CT X+2 Y+30 CCA90 R+5 RL F250
8  FC DR- R18 CLSD+ CCX+20 CCV+30
                                        00:00:00

ACTL.    ⊕X   +250,0000    ⊕Y   +102,3880
         ⊕Z   -114,0914    ⊕C    +30,0000
         ⊕B    +90,0000
T                          F  0       M 5/9

■   ⊟  ▱        RESTORE        /□      TOOL
                POS. AT       OFF/ON   TABLE
                  N
```

```
PROGRAM RUN, FULL SEQUENCE         TEST RUN

0  BEGIN PGM FK1 MM
1  BLK FORM 0.1 Z X+0 Y+0 Z-20         NOML.
2  BLK FORM 0.2 X+100 Y+100 Z+0        X  +250.0000
3  TOOL CALL 1 Z S500                  Y  +102.3880
4  L Z+250 R0 F MAX                    Z  -114.0914
5  L X-20 Y+30 R0 F MAX                C   +30.0000
6  L Z-10 R0 F1000 M3                  B   +90.0000
7  APPR CT X+2 Y+30 CCA90 R+5 RL F250
8  FC DR- R18 CLSD+ CCX+20 CCV+30      ◹        B +90.0000
                                                C +30.0000

                                       ◹  BASIC ROTATION  +0.0000

ACTL.    ⊕X   +250,0000    ⊕Y   +102,3880
         ⊕Z   -114,0914    ⊕C    +30,0000
         ⊕B    +90,0000
T                          F  0       M 5/9

PAGE  PAGE   BEGIN  END   RESTORE       /□
 ⇧    ⇩      TEXT   TEXT  POS. AT      OFF/ON
                             N
```

You can choose between several additional status displays with the following soft keys:

▷ Shift the soft-key rows until the STATUS soft keys appear.

| STATUS PGM | Select the desired additional status display, e.g. general program information.

| STATUS PGM | ## General program information

1 Name of main program

2 Active programs

3 Active machining cycle

4 Circle center CC (pole)

5 Operating time

6 Dwell time counter

```
1   PROGRAMS        AFK1

2   PGM   AFK2
    CALL

3   CYCL    1  PECKING
    DEF

4   CC  X  +12,5000         DWELL      6
    ⊕   Y  +12,5000

              5   ⊙  00:00:04
```

| STATUS POS. | ## Positions and coordinates

1 Position display

2 Type of position display, e.g. actual positions

3 Tilt angle of the working plane

4 Angle of a basic rotation

```
1   ACTL 2
    X   +114,185
    Y    +82,136
    Z    +48,889
    C   +109,995
    B    +25,000

3                   B  +25,0000
                    C  +90,0000

4       BASIC ROTATION
```

STATUS TOOL

Information on tools

1 T: Tool number and name
RT: Number and name of a replacement tool

2 Tool axis

3 Tool length and radii

4 Oversizes (delta values) from TOOL CALL (PGM) and the tool table (TAB)

5 Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)

6 Display of the active tool and the (next) replacement tool

```
1    TOOL      RT 10
2   Z              3    L      -1,2560
                        R      +5,0000
                        R2     +0,2500
4           DL        DR        DR2
     TAB  +0,1000   +0,1000   +0,0100
     PGM  +0,1000   +0,1000
5        CUR.TIME    TIME1     TIME2
          00:25      04:20     04:10
6    TOOL CALL   1
     RT   ←→
```

STATUS COORD. TRANSF.

Coordinate transformations

1 Name of main program

2 Active datum shift (Cycle 7)

3 Active rotation angle (Cycle 10)

4 Mirrored axes (Cycle 8)

5 Active scaling factor(s) (Cycles 11 / 26)

6 Scaling datum

See also section 8.7 "Coordinate Transformation Cycles."

```
1    PROGRAMS       AFK1
2    DATUM SHIFT        ROTATION      3
      X   +1,5000        +12,5000
      Y  +10,0000
      Z  -25,0000        MIRROR IMAGE  4
                            X Y
            SCALING
      X   +0,0000        0,999500
      Y   +0,0000        0,999500
6                  5
```

STATUS TOOL PROBE

Tool measurement

1 Number of the tool to be measured

2 Display whether the tool radius or the tool length is being measured

3 MIN and MAX values of the single cutting edges and the result of measuring the rotating tool (DYN)

4 Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance in the tool table was exceeded.

```
1    TOOL     T 1
        R  2  3 MIN 2    +8.4171
                 MAX 1    +8.7554
                 DYN      +8.8964
4    1   +8.7554 *
     2   +8.4171 *
     3   +8.7293 *
     4   +8.7464 *
```

1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D Touch Probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run
- Digitize 3-D surfaces (option), and
- Measure and inspect tools

TS 220 and TS 630 triggering touch probes
These touch probes are particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 630 features infrared transmission of the triggering signal to the TNC. This makes it highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear resisting optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the TNC, which stores the current position of the stylus as an actual value.

During digitizing the TNC generates a program containing straight line blocks in HEIDENHAIN format from a series of measured position data. You can then output the program to a PC for further processing with the SUSA evaluation software. This evaluation software enables you to calculate male/female transformations or correct the program to account for special tool shapes and radii that differ from the shape of the stylus tip. If the tool has the same radius as the stylus tip you can run these programs immediately.

TT 120 tool touch probe for tool measurement
The TT 120 is a triggering 3-D touch probe for tool measurement and inspection. Your TNC provides three cycles for this touch probe with which you can measure the tool length and radius automatically — either with the spindle rotating or stopped.

The TT 120 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.

HR electronic handwheels
Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 integral handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.

2

Manual Operation and Setup

2.1 Switch-On

⚒ Switch-on and traversing the reference points can vary depending on the individual machine tool. Your machine manual provides more detailed information.

▶ Switch on the power supply for control and machine.

The TNC automatically initiates the following dialog:

MEMORY TEST

The TNC memory is automatically checked.

POWER INTERRUPTED

CE TNC message that the power was interrupted — clear the message.

TRANSLATE PLC PROGRAM

The PLC program of the TNC is automatically translated.

RELAY EXT. DC VOLTAGE MISSING

I Switch on the control voltage.
The TNC checks the functioning of the EMERGENCY STOP circuit.

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

I Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or

X **Y** Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed.

The TNC is now ready for operation in the MANUAL OPERATION mode.

☞ The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the PROGRAMMING AND EDITING or TEST RUN modes of operation immediately after switching on the control voltage.

You can then traverse the reference points later by pressing the PASS OVER REFERENCE soft key in the MANUAL OPERATION mode.

Traversing the reference point in a tilted working plane
The reference point of a tilted coordinate system can be traversed by pressing the machine axis direction buttons. The "tilting the working plane" function (see section 2.5 "Tilting the Working Plane") must be active in the MANUAL OPERATION mode. The TNC then interpolates the corresponding axes.

The NC START button is not effective. Pressing this button may result in an error message.

Make sure that the angle values entered in the menu for tilting the working plane match the actual angle of the tilted axis.

2.2 Moving the Machine Axes

Traversing with the machine axis direction buttons is a machine-dependent function. Your machine manual provides more detailed information.

To traverse with the machine axis direction buttons:

🖐	Select the MANUAL OPERATION mode.

X	Press the machine axis direction button and hold it as long as you wish the axis to move.

...or move the axis continuously:
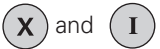
X and **I**	Press and hold the machine axis direction button, then press the machine START button: The axis continues to move after you release the keys.

0	To stop the axis, press the machine STOP button.

You can move several axes at a time with these two methods.

Traversing with the HR 410 electronic handwheel

The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip. You can only move the machine axes when an permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

1 EMERGENCY STOP

2 Handwheel

3 Permissive buttons

4 Axis address keys

5 Actual-position-capture key

6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)

7 Direction in which the TNC moves the selected axis

8 Machine function
(set by the machine tool builder)

The red indicators show the axis and feed rate you have selected.

It is also possible to move the machine axes with the handwheel during a program run.

To move an axis:

🎛	Select the operating mode ELECTRONIC HANDWHEEL.
▬	Press the permissive button.
X	Select the axis.
WW	Select the feed rate.
➕ or ➖	Move the active axis in the positive or negative direction.
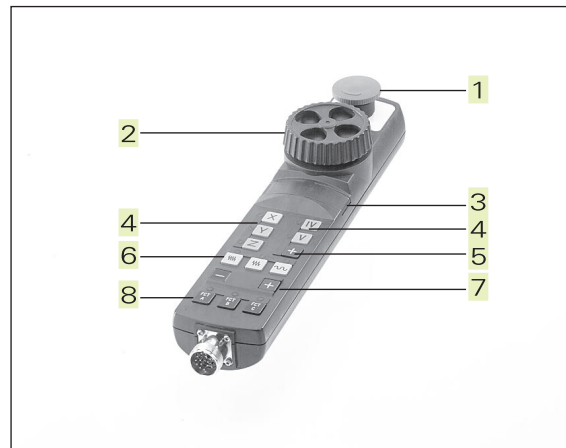
Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance each time you press the corresponding machine axis direction button.

| ⊛ | Select the operating mode ELECTRONIC HANDWHEEL. |

| I | Select incremental jog positioning (the valid key is determined by the machine tool builder). |

JOG INCREMENT =

| 8 ENT | Enter the jog increment in millimeters (here, 8 mm). |

| X | Press the machine axis direction button as often as desired. |

Incremental jog positioning is a machine-dependent function. Your machine manual provides more detailed information.

The machine tool builder determines whether the interpolation factor for each axis is set at the keyboard or with a step switch.
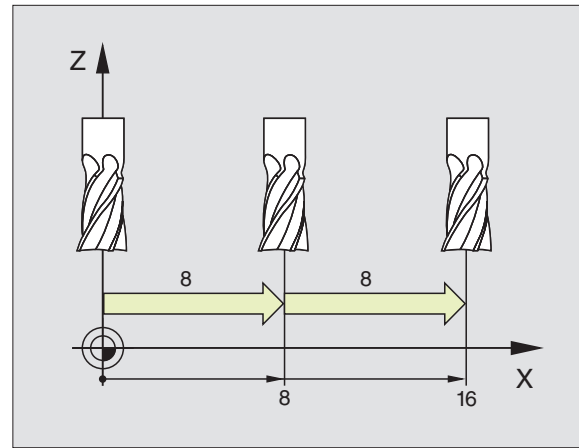
2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M

In the operating modes MANUAL OPERATION and ELECTRONIC HANDWHEEL, you can enter the spindle speed S and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions." The feed rate is defined in a machine parameter and can be changed only with the override knobs (see next page).

Entering values
Example: Enter the spindle speed S

| S | To select the spindle speed, press the S soft key.

`SPINDLE SPEED S=`

1000 [ENT]   Enter the desired spindle speed,

(I)   and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function.

Proceed in the same way to enter the miscellaneous functions M.

Changing the spindle speed S and feed rate F
With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.

The knob for spindle speed override is effective only on machines with a stepless spindle drive.

The machine tool builder determines which miscellaneous functions M are available on your TNC and what effects they have.
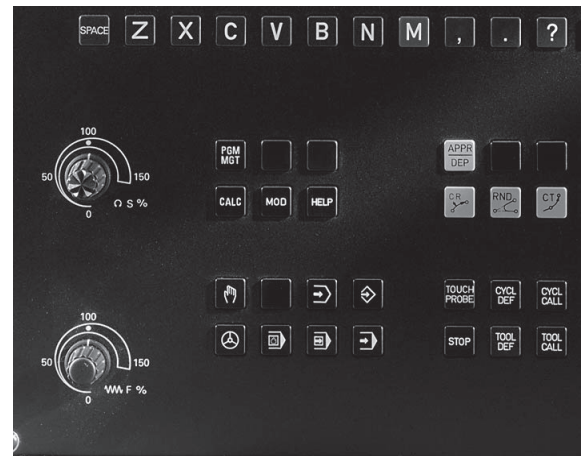
2.4 Setting the Datum (Without a 3-D Touch Probe)

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

To prepare the TNC:

▶ Clamp and align the workpiece.

▶ Insert the zero tool with known radius into the spindle.

▶ Ensure that the TNC is showing the actual position values.

Setting the datum
Fragile workpiece? If the workpiece surface must not be scratched, you can lay a metal shim of know thickness d on it. Then enter a tool axis datum value that is larger than the desired datum by the value d.

🖐	Select the MANUAL OPERATION mode.
⊗ X	Move the tool slowly until it touches the workpiece surface.
X	Select the axis.

DATUM   SET   X =

0 ENT	Zero tool: Set the display to a known workpiece position (here, 0) or enter the thickness *d* of the shim.

Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

2.5   Tilting the Working Plane

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With specific swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as solid angles. Your machine manual provides more detailed information on this subject.

The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are two ways to tilt the working plane:

■ 3D ROT soft key in the MANUAL OPERATION and ELECTRONIC HANDWHEEL operating modes (described below)

■ Cycle 19 WORKING PLANE in the part program (see page 200).
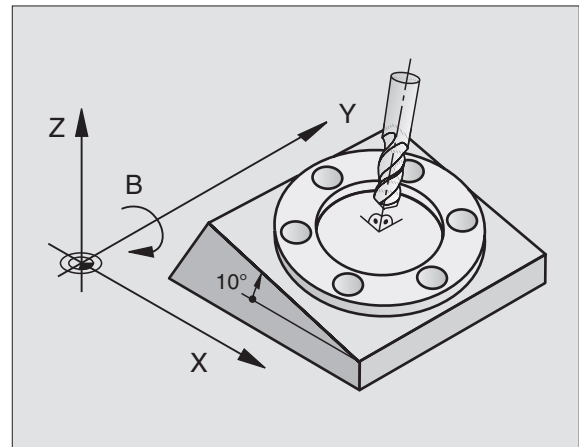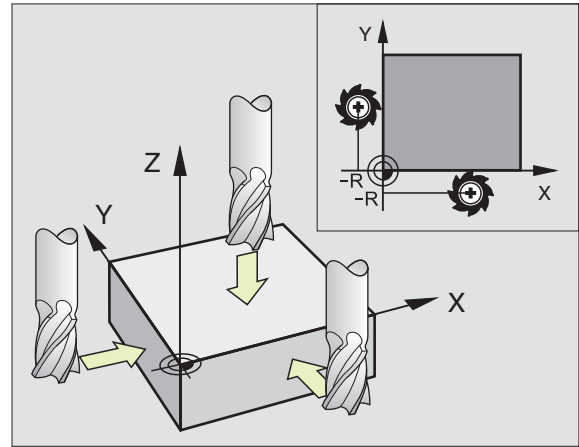
The TNC functions for "tilting the working plane" are coordinate transformations in which the working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types:

Machines with tilting tables:
■ You must tilt the **workpiece** into the desired position for machining by positioning the tilting table, for example with an L block.

■ The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table — and therefore the workpiece — by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in Z+ direction.

■ In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).

Machines with swivel heads
■ You must bring the **tool** into the desired position for machining by positioning the swivel head, for example with an L block.

■ The position of the transformed tool axis **changes** in relation to the machine-based coordinate system. Thus if you rotate the swivel head — and therefore the tool — in the B axis by 90° for example, **the coordinate system rotates also**. If you press the Z+ axis direction button in the MANUAL OPERATION mode, the tool moves in X+ direction of the machine-based coordinate system.

■ In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) **and** offsets caused by tilting of the tool (3-D tool length compensation).

Traversing the reference points in tilted axes
With tilted axes, you use the machine axis direction buttons to cross over the reference points. The TNC interpolates the corresponding axes. Be sure that the function for tilting the working plane is active in the MANUAL OPERATION mode and the actual angle of the tilted axis was entered in the menu field.

Setting the datum in a tilted coordinate system
After you have positioned the tilted axes, set the datum in the same way as for non-tilted axes. The TNC then converts the datum for the tilted coordinate system. If your machine tool features axis control, the angular values for this calculation are taken from the actual position of the tilted axis.

☞ If your machine tool is not equipped with axis control, you must enter the actual position of the tilted axis in the menu for manual tilting: The actual positions of one or several tilted axes must match the entry. Otherwise the TNC will calculate an incorrect datum.

Position display in a tilted system
The positions displayed in the status window (ACTL. and NOML.) are referenced to the tilted coordinate system.

Limitations on working with the tilting function
■ The touch probe function BASIC ROTATION cannot be used.

■ PLC positioning (determined by the machine tool builder) is not possible.

■ Positioning blocks with M91/M92 are not permitted.

To activate manual tilting:

To select manual tilting, press the 3D ROT soft key.
You can now select the desired menu option with the arrow keys.

Enter the tilt angle.

To set the desired operating mode in menu option TILT WORK-ING PLANE to ACTIVE, select the menu option and shift with the ENT key.

To conclude entry, press the END soft key.

To reset the tilting function, set the desired operating modes in menu TILT WORKING PLANE to INACTIVE.

Whenever the function TILT WORKING PLANE is active and the TNC is moving the machine axes in the tilted plane, the symbol ⌖ is shown in the status display.

If you set the function TILT WORKING PLANE for the operating mode PROGRAM RUN to ACTIVE, the tilt angle entered in the menu becomes active in the first block of the part program. If you are using Cycle 19 WORKING PLANE in the part program, the angular values defined in the cycle (starting at the cycle definition) are effective. Angular values entered in the menu will be overwritten.

```
MANUAL OPERATION              PROGRAMMING
                              AND EDITING

TILT WORKING PLANE
PROGRAM RUN:              ACTIVE
MANUAL OPERATION         INACTIVE


B = +90          °
C = +30          °


ACTL.   +X  +250,0000    +Y   +102,3880
        +Z  -114,0914    +C    +30,0000
        +B   +90,0000
T                        F 0       M 5/9
```

3

Positioning with
Manual Data Input (MDI)

3.1 Programming and Executing Simple Machining Operations

The operating mode POSITIONING WITH MANUAL DATA INPUT is particularly convenient for simple machining operations or pre-positioning of the tool. It enables you to write a short program in HEIDENHAIN conversational programming or in ISO format, and execute it immediately. You can also call TNC cycles. The program is stored in the file $MDI. In the operating mode POSITIONING WITH MDI, the additional status displays can also be activated.

🖐   Select the operating mode POSITIONING WITH MDI.
Program the $MDI file as desired.

Ⓘ   To start program run, press the machine START button.

☞   Limitation: FK free contour programming, programming graphics and program run graphics cannot be used. The $MDI file must not contain a program call (PGM CALL).

Example 1
A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool in L blocks (straight-line blocks) to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle 1 PECKING.
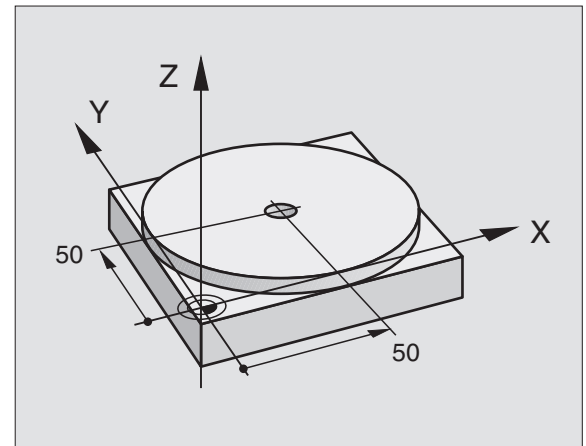
`0  BEGIN PGM $MDI MM`	
`1  TOOL DEF 1 L+0 R+5`	Define tool: zero tool, radius 5
`2  TOOL CALL 1 Z S2000`	Call tool: Tool axis Z, spindle speed 2000 rpm
`3  L Z+200 R0 F MAX`	Retract tool (F MAX = rapid traverse)
`4  L X+50 Y+50 R0 F MAX M3`	Position tool at FMAX above hole, spindle ON
`5  L Z+5 F2000`	Position tool to 5 mm above hole

6 CYCL DEF 1.0 PECKING	Define PECKING cycle:
7 CYCL DEF 1.1 SET UP 5	Setup clearance of the tool above the hole
8 CYCL DEF 1.2 DEPTH -20	Total hole depth (Algebraic sign=working direction)
9 CYCL DEF 1.3 PECKG 10	Depth of each infeed before retraction
10 CYCL DEF 1.4 DWELL 0.5	Dwell time in seconds at the hole bottom
11 CYCL DEF 1.5 F250	Feed rate for pecking
12 CYCL CALL	Call PECKING cycle
13 L Z+200 R0 F MAX M2	Retract tool
14 END PGM $MDI MM	End of program

The straight-line function is described in section 6.4 "Path Contours — Cartesian Coordinates," the PECKING cycle in section 8.2 "Drilling Cycles."

Example 2
Correcting workpiece misalignment on machines with rotary tables

Make a basic rotation with the 3-D touch probe as described in section 12.2 "Touch Probe Cycles in the MANUAL and ELECTRONIC HANDWHEEL Operating Modes," under the section "Compensating workpiece misalignment."

Write down the ROTATION ANGLE and cancel the BASIC ROTATION.

Select the POSITIONING WITH MDI mode of operation.

IV Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate.
For example: L C+2.561 F50

Conclude entry.

I Press the machine START button: The rotation of the table corrects the misalignment.

Protecting and erasing programs in $MDI

The $MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:

◈   Select the PROGRAMMING AND EDITING mode of operation.

PGM MGT   To call the file manager, press the PGM MGT key (program management).

↑   Move the highlight to the $MDI file.

COPY ABC⇒XYZ   To select the file copying function, press the COPY soft key.

DESTINATION FILE =

BOREHOLE   Enter the name under which you want to save the current contents of the $MDI file.

EXECUTE   Copy the file.

END   To close the file manager, press the END soft key.

Erasing the contents of the $MDI file is done in a similar way: Instead of copying the contents, however, you erase them with the DELETE soft key. The next time you select the operating mode POSITIONING WITH MDI, the TNC will display an empty $MDI file.

For further information, refer to section 4.2 "File Management."

4

Programming

**Fundamentals of NC,
File Management,
Programming Aids**

4.1 Fundamentals of NC

Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The TNC evaluates this signal and calculates the precise actual position of the machine axis.

If there is an interruption of power, the calculated position will no longer correspond to the actual position of the machine slide. The TNC can re-establish this relationship with the aid of reference marks when power is returned. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From the signal the TNC identifies that position as the machine-axis reference point and can re-establish the assignment of displayed positions to machine axis positions.

Linear encoders are generally used for linear axes. Rotary tables and tilt axes have angle encoders. If the position encoders feature distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders, to re-establish the assignment of the displayed positions to machine axis positions.
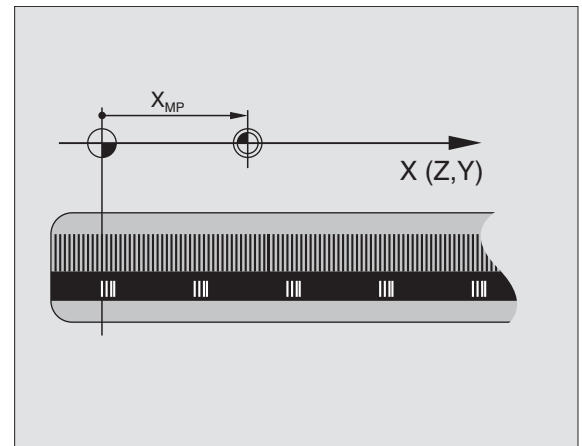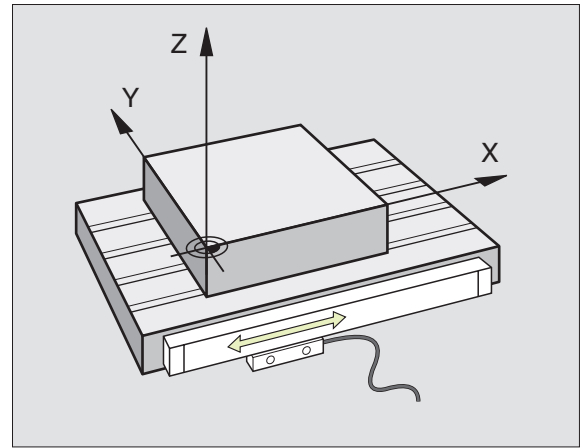
Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also refered to as incremental coordinate values.
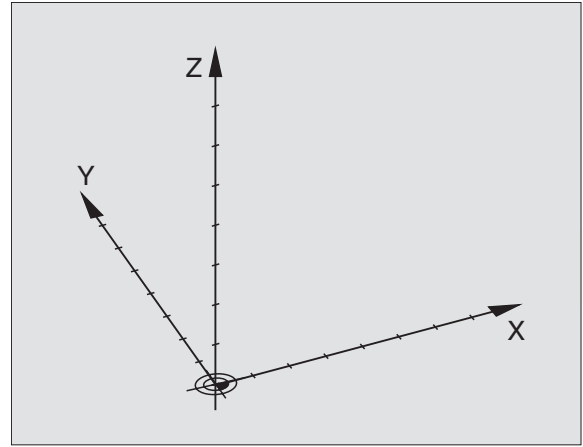
Reference systems on milling machines

On a milling machine, workpieces are normally machined according to the Cartesian coordinate system that is parallel to the machine axes. The figure at right illustrates the "right-hand rule" for remembering the three axis directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.

The TNC 426 can control the machine tool in up to 5 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The figure to the lower right shows the main axes and their associated secondary and rotary axes.
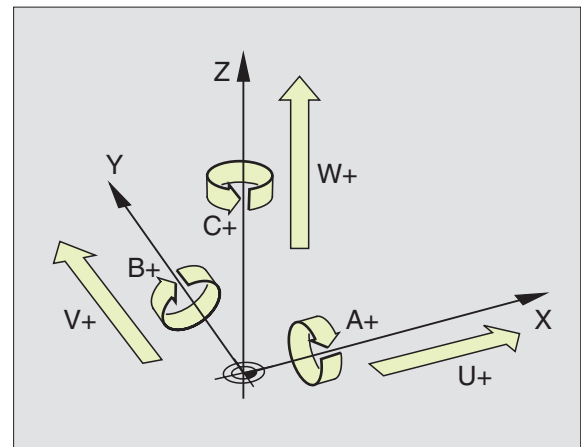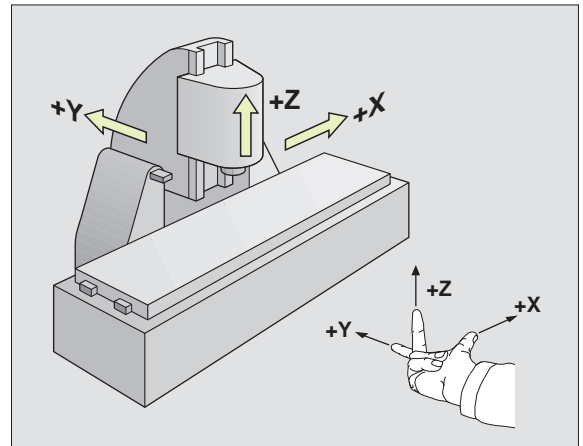
Polar coordinates

If the production drawing is dimensioned in Cartesian coordinates, you also write the part program using Cartesian coordinates. For parts containing circular arcs or angles it is often simpler to give the dimensions in polar coordinates.

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the

■ Polar Radius, the distance from the circle center CC to the position, and the

■ Polar Angle, the size of the angle between the reference axis and the line that connects the circle center CC with the position.

See figure to the lower right.

Setting the pole and the angle reference axis
The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle PA.

Coordinates of the pole (plane)	Reference axis of the angle
XY	+X
YZ	+Y
ZX	+Z
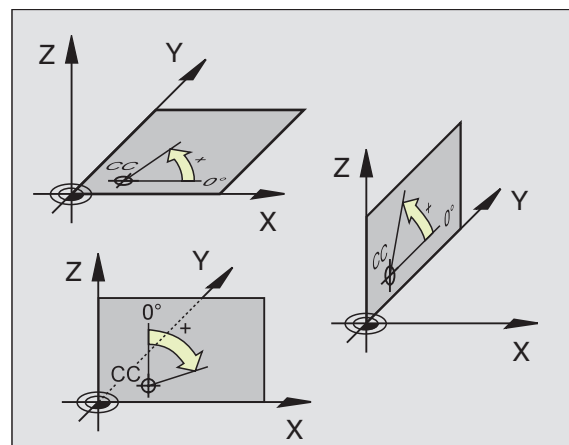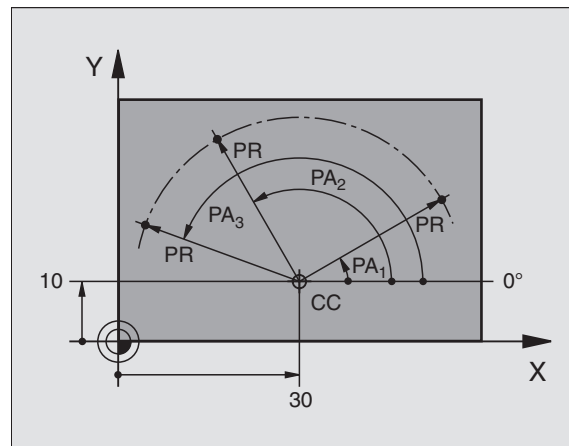
Absolute and relative workpiece positions

Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is uniquely defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

Hole 1	Hole 2	Hole 3
X=10 mm	X=30 mm	X=50 mm
Y=10 mm	Y=20 mm	Y=30 mm
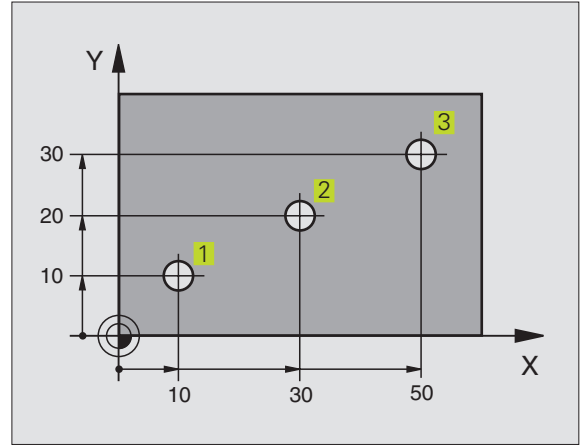
Relative workpiece positions

Relative coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write a part program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. Incremental coordinates are therefore also referred to as chain dimensions.

To program a position in incremental coordinates, enter the prefix "I" before the axis.

Example 2: Holes dimensioned in relative coordinates

Absolute coordinates of hole 4:

X= 10 mm
Y= 10 mm

Hole 5 referenced to hole 4	Hole 6 referenced to hole 5
IX= 20 mm	IX= 20 mm
IY= 10 mm	IY= 10 mm
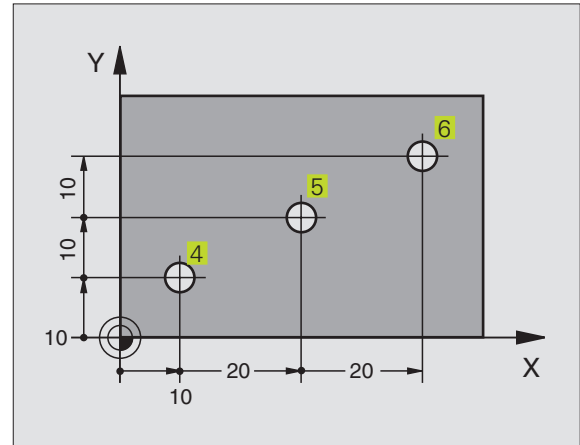
Absolute and incremental polar coordinates

Absolute polar coordinates always refer to the pole and the reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.
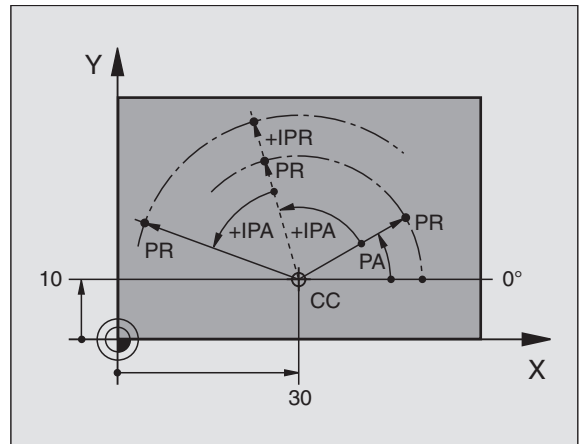
Selecting the datum

A production drawing identifies a certain point on the workpiece — usually a corner — as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display to either zero or a predetermined position value. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.

If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles. For further information, refer to section 8.7 "Coordinate Transformation Cycles."

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece, which is the most suitable for deducing the dimensions of the remaining workpiece positions.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. For further information, refer to section 12.2 "Setting the Datum with a 3-D Touch Probe."

Example

The workpiece drawing at right illustrates the holes 1 to 4, which are dimensioned to an absolute datum with the coordinates X=0 Y=0. The holes 5 to 7 are referenced to a relative datum with the absolute coordinates X=450 Y=750. By using the DATUM SHIFT cycle you can shift the datum temporarily to the position X=450, Y=750 and program the holes 5 to 7 without any further calculations.
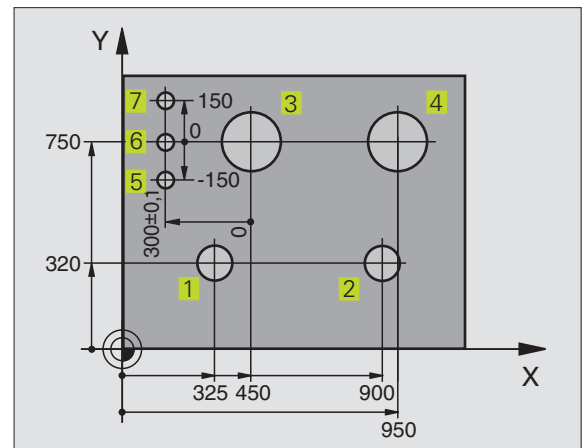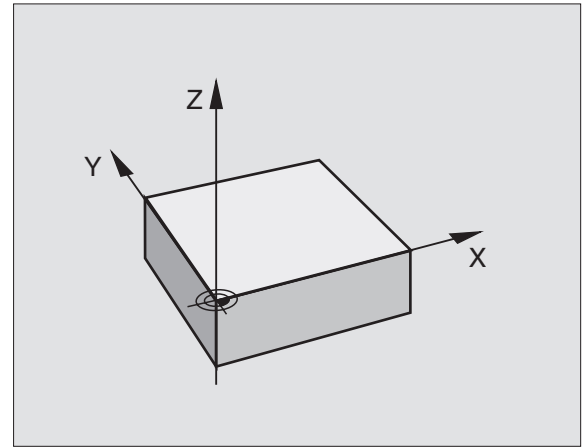
4.2 File Management

Files and file management

When you write a part program on the TNC, you must first enter a file name. The TNC then stores the program on the hard disk as a file with this name. You can also store texts and tables as files.

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. Directories also have names and work like the drawers of a filing cabinet: they enable you to save your files in groups according to some system. You could set up your directories according to job number, for example. If you divide a directory up into further directories, these subordinate directories are called subdirectories.

The TNC provides a special file management window in which you can easily find and manage your files. You can call, copy, rename and erase any of your files, or create, copy and erase directories.

File names and directory names
The name of a file or directory can have up to 8 characters. When you store programs, tables and texts as files, the TNC adds an extension to the file name, separated by a point. This extension identifies the file type (see table at right).

PROG20	.H
File name	File type
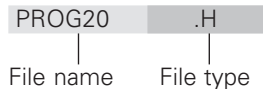
Directories are created in the file management window. The name of a directory can also contain up to 8 characters and does not have an extension.

You can manage any number of files on the TNC's hard disk. Their total size, however, must not exceed 170 MB. If you save more than 512 files in one directory, the TNC no longer sorts them alphabetically.

Data security

We recommend saving newly written programs and files on a PC at regular intervals. You can do this with the cost-free backup program TNCBACK.EXE from HEIDENHAIN.

In addition, you need a floppy disk on which all machine-specific data, such as PLC program, machine parameters, etc., are stored. Please contact your machine tool builder for more information on both the backup program and the floppy disk.

Files in the TNC	Type
Programs	
in HEIDENHAIN conversational format	.H
in ISO format	.I
Tables for	
Tools	.T
Pallets	.P
Datums	.D
Points (digitizing range for measuring touch probe)	.PNT
Texts as	
ASCII files	.A

☞ Saving the entire disk (up to 170 MB) may take several hours. In this case, it is a good idea to save the data outside of work hours, e.g. during the night.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by the symbol "\".

Example: On drive TNC:\, the directory AUFTR1 was created. Under this directory, the subdirectory NCPROG was created, and the part program PROG1.H copied into this subdirectory. The part program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.H

The chart at right illustrates an example of a directory display with different paths. This branched structure is often referred to as a "tree." The term "tree" is used in various soft keys on the TNC.
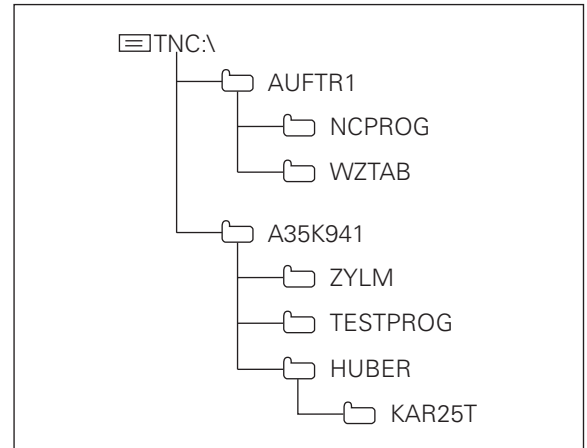
Working with the file manager

This section informs you about the two screen layout options for file management, the meaning of the individual screen information, and describes how to select files and directories. If you are not yet familiar with the TNC file manager, we recommend that you read this section completely and test the individual functions on your TNC.

To call the file manager

PGM MGT

Press the PGM MGT key:
the TNC displays the file management window

In file management, as in the other operating modes, the TNC always displays the screen layout that you last selected. If the layout on your TNC screen does not look the same as the figure at right, you can change it with the WINDOW soft key.

The screen layout shown in the figure at right is particularly convenient for calling or renaming programs, and creating directories.

The narrow window on the left indicates three drives **1** on the top of the screen. Drives designate devices with which data are stored or transferred. One drive is the hard disk of the TNC. Other drives are the interfaces (RS232, RS422), which can be used, for example, for connecting a personal computer. The selected (active) drive is shown in a different color.

In the lower part of the narrow screen, the TNC displays all directories **2** stored on the selected drive. A drive is always identified by a file symbol to the left and the directory name to the right. The TNC displays a subdirectory to the right of and below its parent directory. The selected (active) directory is depicted in a different color.

The large window on the right displays all files **3** that are stored in the selected directory. Each file is shown with additional information that is illustrated in the table on the next page.

To select drives, directories and files:

PGM MGT	Call the file manager.

Use the arrow keys to move the highlight to the desired position on the screen:

↑ ↓	Move the highlight up and down within a window
← →	Move the highlight from the left to the right window, and vice versa.

Select a drive first:

Move the highlight to the desired drive in the left window:

SELECT or ENT	To select the drive, press the SELECT soft key or ENT key.

Then select a directory:

Move the highlight to the desired directory in the left window — the right window shows all files stored in the highlighted directory.

Select a file, or create a new directory as described on the next page.

Display	Meaning
FILE NAME	Name with up to 8 characters and file type
BYTE	File size in bytes
STATUS	Properties of the file:
E	Program is selected in the PROGRAMMING AND EDITING mode of operation
S	Program is selected in the TEST RUN mode of operation
M	Program is selected in a program run mode of operation
P	File is protected against editing and erasure
IN	File contains inch dimensions
W	File was incompletely transferred to external storage (write error)
DATE	Date the file was last changed
TIME	Time the file was last changed

4.2 File Management

To select a file:

Move the highlight to the desired file in the right window:

Display of long file directories	Soft key
Move pagewise up through the file directory	PAGE ⇧
Move pagewise down through the file directory	PAGE ⇩

SELECT or ENT — To open a file, press the SELECT soft key or ENT key. The selected file is opened in the operating mode from which you have call the file manager.

To create a new directory (only possible in the TNC's hard disk drive):

Move the highlight in the left window to the directory in which you want to create a subdirectory.

NEW ENT — Enter the new file name, and confirm with ENT.

DIRECTORY \CREATE NEW ?

YES — Confirm with the YES soft key, or

NO — Abort with the NO soft key.

Additional file management functions are described starting from "Overview: Additional file functions" on page 36.

To copy directories and files as well as for selecting data transmission to a PC, select the screen layout with the two equally sized windows (see figure at right):

WINDOW
≡|≡ ≡

To change the screen layout, press the WINDOW soft key.

In this layout option, the TNC displays only files or only directories in a window.

If the TNC displays a window containing files, the soft key PATH appears in the soft-key row. "PATH" stands for the structure of the directories.

PATH

To display directories, press the PATH soft key.

If the TNC displays a window containing directories, the soft key FILES appears in the soft-key row:

FILES

To display files, press the FILES soft key.

With the arrow keys, you can move the highlight to the desired position on the screen.

PROGRAM RUN
FULL SEQUENCE

PROGRAM TABLE EDITING
FILE NAME = SDG.A

TNC:\NK\KLART*.*

FILE NAME		BYTES	STATUS
SDG	.A	1	
3500	.H	1142	
3501	.H	518	
3503	.H	1294	
3504	.H	1106	
3506	.H	756	
3507	.H	1220	
3508	.H	1490	
3510	.H	1222	
3511	.H	2216	
3513	.H	952	

15 FILE(S) 170496 KBYTE VACANT

TNC:*.*

FILE NAME		BYTES	STATUS
1234	.	1	
%TCHPRNT	.A	398	
CVREPORT	.A	8903	
WINK	.BAT	22	
WINK_TNC	.BAT	22	
$MDI	.H	186	
123	.H	468	
1234	.H	96	
12345	.H	12	
79138	.H	2678	
BABV	.H	18586	

21 FILE(S) 170496 KBYTE VACANT

PAGE ⇑ PAGE ⇓ SELECT COPY ABC⇒XYZ SELECT TYPE WINDOW ≡≡≡ PATH END

To select a drive:

PATH

If the selected window does not display directories, press the PATH soft key.

SELECT or ENT

Move the highlight to the desired drive and confirm with the SELECT soft key or ENT key: The window displays the files stored in this drive.

To select a directory:

PATH

Press the PATH soft key.

SELECT or ENT

Move the highlight to the desired directory and confirm with the SELECT soft key or ENT key: The window displays the files stored in this directory.

To select a file:

SELECT or ENT

Move the highlight to the desired file and confirm with the SELECT soft key or ENT key: The selected file is opened in the operating mode from which you have the called file manager.

Overview: Additional file functions

This table provides an overview of the functions described on the following pages:

Function	Soft key
Display a specific file type	SELECT TYPE
Copy (and convert) a file	COPY ABC⇒XYZ
Display the last 10 files that were selected	LAST FILES
Erase a file or directory	DELETE
Rename a file	RENAME ABC = XYZ
Tag a file	TAG
Protect a file against editing and erasure	PROTECT
Cancel file protection	UNPROTECT
Convert an FK program into HEIDENHAIN conversational format	CONVERT FK->H
Copy a directory	COPY DIR
Erase a directory together with all its subdirectories	DELETE ALL
Display the directories stored on an external drive	SHOW TREE
Select a directory from an external drive	SELECT PATH

Displaying a specific file type
Displaying all file types

SELECT TYPE	Press the SELECT TYPE soft key.
SHOW .A	Press the soft key for the desired file type, or
SHOW ALL	Press the SHOW ALL soft key to display all files.

Copying individual files

▶ Move the highlight to the file you wish to copy.

COPY ABC⇒XYZ	▶ Press the COPY soft key to select the copying function.

▶ Enter the name of the destination file and confirm your entry with the ENT key or EXECUTE soft key: The TNC copies the file into the active directory. The original file is retained.

Copying a directory

If you wish to copy a directory including all its subdirectories, press the COPY DIR soft key instead of the COPY soft key.

Copying a table

If you are copying tables, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:

■ The target table must exist.

■ The file to be copied must only contain the columns or lines you want to replace.

Erasing files

▶ Move the highlight to the file you want to erase, or tag several files (see "Tagging files").

DELETE	▶ To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the files.
	▶ To confirm erasure press the YES soft key. Abort with the NO soft key if you do not wish to erase the files.

Erasing directories

▶ Erase all files stored in the directory you wish to erase.

▶ Move the highlight to the directory.

DELETE	▶ Select the erasing function.
	▶ To confirm erasure, press the YES soft key.
	Abort with the NO soft key if you do not wish to erase the directory.

Renaming a file

▶ Move the highlight to the file you wish to rename.

RENAME ABC = XYZ	▶ Select the renaming function.
	▶ Enter the new file name; the file type cannot be changed.
	▶ To execute renaming, press the ENT key.

Tagging files

Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files, proceed as follows:

Move the highlight to the first file.

| TAG | To display the tagging functions, press the TAG soft key. |

| TAG FILE | To tag the file, press the TAG FILE soft key. |

Move the highlight to the next file you wish to tag:

| TAG FILE | To select the next file, press the TAG FILE soft key. Repeat this process for all files you want to tag. |

Additional tagging functions	Soft key
Tag all files in the directory	TAG ALL FILES
Untag one file	UNTAG FILE
Untag all files	UNTAG ALL FILES

Copying files into another directory

▶ Select the screen layout with the two equally sized windows.

▶ To display directories in both windows, press the PATH soft key.

In the right window:
  ▶ Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key.

In the left window:
  ▶ Move the highlight to the directory containing the files that you want to copy.

TAG	▶ Display the file tagging functions.
TAG FILE	▶ Move the highlight to the file you want to copy and tag it. You can tag several files in this way, as desired.
COPY TAG	▶ Copy the tagged files into the target directory.

For further tagging functions, see "Tagging files" above.

You can also copy files if you have selected a screen layout with a small window on the left and a large window on the right. Tag the files you want to copy in the right window with the TAG FILE or TAG ALL FILES soft key. When you press COPY TAG, the TNC asks you to enter the target directory: Type in the complete path name, including the drive.

Overwriting files

If you copy files into a directory in which other files are stored under the same name, the TNC will ask whether the files in the target directory should be overwritten:

▶ To overwrite all files, press the YES soft key, or

▶ To overwrite no files, press the NO soft key, or

▶ To confirm of each file separately before overwriting it, press the CONFIRM soft key.

A protected file cannot be overwritten. You must remove file protection first.

Protecting a file/Canceling file protection

▶ Move the highlight to the file you want to protect.

| MORE FUNCTIONS | ▶ To select the additional functions, press the MORE FUNCTIONS soft key. |

| PROTECT | ▶ To enable file protection, press the PROTECT soft key. The file now has status P. |

To cancel file protection, proceed in the same way using the UNPROTECT soft key.

Converting individual files

▶ Move the highlight to the file you want to convert.

COPY ABC⇒XYZ	▶ Press the COPY soft key.
	▶ Type the name of the destination file in the screen headline and indicate the desired file type — separated by a point.
	▶ Confirm your entry with the EXECUTE soft key or ENT key.

Converting several files

▶ Tag several files with the TAG FILE or TAG ALL FILES soft key.

COPY TAG	▶ Press the COPY TAG soft key.
	▶ Instead of a file name, enter the wildcard symbol "*" into the screen headline and indicate the desired file type — separated by a point.
	▶ Confirm your entry with the EXECUTE soft key or ENT key.

Converting an FK program into HEIDENHAIN conversational format

▶ Move the highlight to the file you want to convert.

| MORE FUNCTIONS | ▶ To select the additional functions, press the MORE FUNCTIONS soft key. |

CONVERT FK->H	▶ To select the function for converting the file, press the CONVERT FK->H soft key.
	▶ Enter the name of the destination file.
	▶ To execute conversion, press the ENT key.

4.3 Creating and Writing Programs

Organization of an NC program in HEIDENHAIN conversational format

A part program consist of a series of program blocks. The figure at right illustrates the elements of a block.

The TNC numbers the blocks in ascending sequence.

The first block of a program is identified by "BEGIN PGM," the program name and the active unit of measure.

The subsequent blocks contain information on:

- the blank form
- tool definitions and tool calls,
- feed rates and spindle speeds as well as
- path contours, cycles and other functions.

The last block of a program is identified by "END PGM," the program name and the active unit of measure.

Block:

```
10 L X+10 Y+5 R0 F100 M3
```

Path function   Words

Block number
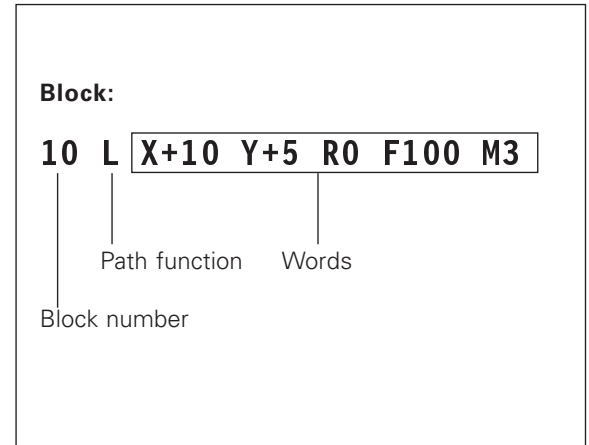
Defining the blank form — BLK FORM

Immediately after initiating a new program, you define a rectangular workpiece blank. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and can be up to 100 000 mm long. The blank form is defined by two of its corner points:

- MIN point: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values.
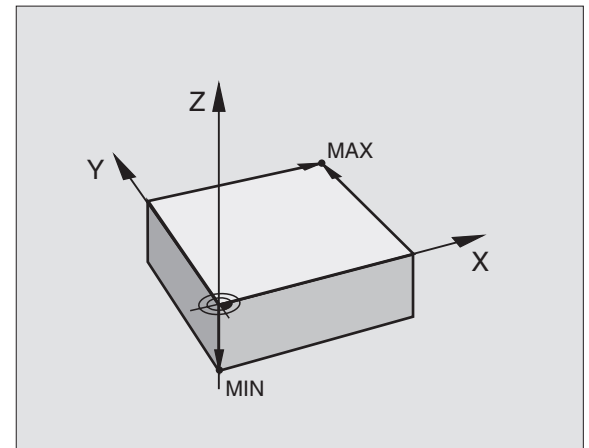
Creating a new part program

You always enter a part program in the PROGRAMMING AND EDITING mode of operation.

Program initiation in an example:

Select the PROGRAMMING AND EDITING mode of operation.

PGM MGT To call the file manager, press the PGM MGT key.

Select the directory in which you wish to store the new program:

FILE NAME=OLD.H

NEW **ENT** Enter the new program name and confirm your entry with the ENT key.

MM To select the unit of measure, press the MM or INCH soft key. The TNC switches the screen layout and initiates the dialog for defining the BLK FORM.

WORKING SPINDLE AXIS X/Y/Z ?

Z Enter the spindle axis.

DEF BLK FORM: MIN-CORNER?

0 **ENT**
0 **ENT**
-40 **ENT**
Enter in sequence the X, Y and Z coordinates of the MIN point.

DEF BLK FORM: MAX-CORNER?

100 **ENT**
100 **ENT**
0 **ENT**
Enter in sequence the X, Y and Z coordinates of the MAX point.

```
MANUAL        PROGRAMMING AND EDITING
OPERATION     DEF BLK FORM: MAX-CORNER ?

0  BEGIN PGM BLK MM
1  BLK FORM 0.1 Z X+0 Y+0 Z-40
2  BLK FORM 0.2 X+100 Y+100
      Z+0
3  END PGM BLK MM
```

The program blocks window shows the following BLK FORM defini-tion:

`0 BEGIN PGM NEU MM`	Program begin, name, unit of measure
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Tool axis, MIN point coordinates
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	MAX point coordinates
`3 END PGM NEU MM`	Program end, name, unit of measure

The TNC automatically generates the block numbers as well as the BEGIN and END blocks.

Programming tool movements in HEIDENHAIN conversational format

To program a block, initiate the dialog by pressing a function key. In the screen headline, the TNC then asks you for all the information necessary to program the desired function.

Dialog initiation in an example

[icon]	Initiate the dialog.

`COORDINATES ?`

X 10	Enter the target coordinate for the X axis.

Y 5 ENT	Enter the target coordinate for the Y axis, and go to the next question with ENT.

`RADIUS COMP. RL/RR/NO COMP. ?`

ENT	Enter "No radius compensation" and go to the next question with ENT.

`FEED RATE F=? / F MAX = ENT`

100 ENT	Enter a feed rate of 100 mm/min for this path contour; go to the next question with ENT.

`MISCELLANEOUS FUNCTION M ?`

3 ENT	Enter the miscellaneous function M3 "spindle ON"; pressing the ENT key will terminate this dialog.

The program blocks window will display the following line:

`3 L X+10 Y+5 R0 F100 M3`

```
MANUAL       PROGRAMMING AND EDITING
OPERATION    MISCELLANEOUS FUNCTION M ?

3   TOOL CALL 1 Z S3150
4   L Z+250 R0 F MAX
5   L X-20 Y+50 R0 F MAX
6   L Z-5 R0 F2000
    L X+10 Y+5 RL F100 M3
7   END PGM NEU MM
```
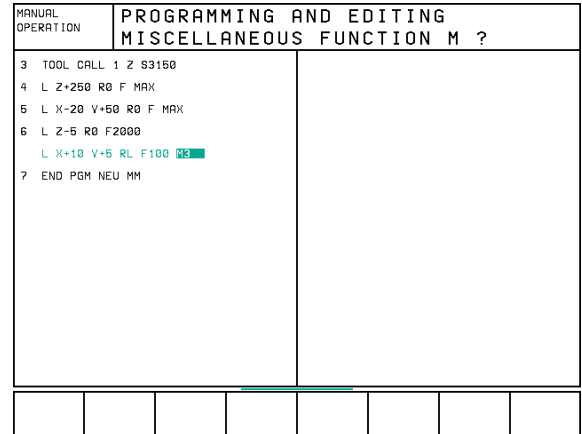
Functions during the dialog	**Key**
Ignore the dialog question	NO ENT
End the dialog immediately	END
Abort the dialog and erase the block	DEL

Editing program lines

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys (see table at right).

Looking for the same words in different blocks

Set the AUTO DRAW soft key to OFF when using this function.

	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word.

	Select a block with the arrow keys.

The word that is highlighted in the new block is the same as the one you selected previously.

Inserting blocks at any desired location

▶ Select the block after which you want to insert a new block and initiate the dialog.

Editing and inserting words

▶ Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.

▶ To conclude editing, press the END key.

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Selecting blocks or words	Keys
Move from one block to the next	
Select individual words in a block	

Erasing blocks and words	Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL
Erase cycles and program sections: First select the last block of the cycle or program section to be erased, then erase with the DEL key.	DEL

4.3 Creating and Writing Programs

4.4 Interactive Programming Graphics

While you are writing the part program, you can have the TNC generate a graphic illustration of the programmed contour.

To generate/not generate graphics during programming:

▶ To switch the screen layout to displaying program blocks to the left and graphics to the right, press the SPLIT SCREEN key and PGM + GRAPHICS soft key.

▶ Set the AUTO DRAW soft key to ON. While you are entering the program lines, the TNC generates each path contour you program in the graphics window in the right screen half.

If you do not wish to have graphics generated during programming, set the AUTO DRAW soft key to OFF.

AUTO DRAW ON does not simulate program section repeats.

To generate a graphic for an existing program:

▶ Use the arrow keys to select the block up to which you want the graphic to be generated, or press GOTO and enter the desired block number.

▶ To generate graphics, press the RESET + START soft key.

Additional functions are listed in the table at right.

Block number display ON/OFF

▶ Shift the soft-key row (see figure at right)

▶ To show block numbers:
Set the SHOW OMIT BLOCK NR. soft key to SHOW.

▶ To omit block numbers:
Set the SHOW OMIT BLOCK NR. soft key to OMIT.

To erase the graphic:

▶ Shift the soft-key row (see figure at right)

▶ To erase the graphic, press the CLEAR GRAPHIC soft key.

Functions	Soft key
Generate interactive graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Interrupt interactive graphics This soft key only appears while the TNC generates the interactive graphics	STOP

Magnifying or reducing a detail

You can select the graphics display by selecting a detail with the frame overlay. You can now magnify or reduce the selected detail.

▶ Select the soft-key row for detail magnification/reduction (second row, see figure at right)
The following functions are available:

Function	Soft key
Show and move the frame overlay. Press and hold the desired soft key to move the frame overlay.	← → ↑ ↓
Reduce the frame overlay — press and hold the soft key to reduce the detail	<<
Enlarge the frame overlay — press and hold the soft key to magnify the detail	>>

```
MANUAL          PROGRAMMING AND EDITING
OPERATION
0  BEGIN PGM 3516 MM
1  BLK FORM 0.1 Z X-90 Y-90 Z-40
2  BLK FORM 0.2 X+90 Y+90 Z+0
3  TOOL DEF 50
4  TOOL CALL 1 Z S1400
5  L Z+50 R0 F MAX
6  L Z+100 R0 F MAX M2
7  LBL 1
8  L X+0 Y+80 RL F250
9  FPOL X+0 Y+0
10 FC DR- R80 CCX+0 CCY+0
11 FCT DR- R7.5
12 FCT DR+ R90 CCX+69.282 CCY-40
13 FSELECT 2 ; VORSCHLAG 1 ENTSPRICHT
   NICHT DER ZEICHNUNG!!!!
```

↑ ↓ ← → << >> WINDOW BLK FORM WINDOW DETAIL

WINDOW DETAIL ▶ Confirm the selected section with the WINDOW DETAIL soft key.

With the WINDOW BLK FORM soft key, you can restore the original section.

4.5 Structuring Programs

This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 244 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex programs in a clear and comprehensible way. This function is particularly convenient if you want to change the program later. Structuring blocks are inserted in the part program. They can also be displayed in a separate window, and edited or added to, as desired. A second level is provided for subdividing a structuring block. The texts in this level are indented.

To display the program structure window / change the active window:

PGM + SECTS ▶ To display the program structure window, select the screen display PGM+SECTS

CHANGE WINDOW ⇨ ▶ The change the active window, press the CHANGE WINDOW soft key.

```
MANUAL          PROGRAMMING AND EDITING
OPERATION
42 CP PA+0 DR- R0 F06          - PARAMETER DEFINITION
43 L Z+08 R0 F9998             - MAKE POCKET
44 FN 1: Q20 = +Q20 + +8.8       - ROUGH OUT
45 FN 9: IF +1 EQU +1 GOTO LBL 111 - FINISHING
46 * - MAKE HOLE PATTERN       - MAKE HOLE PATTERN
47 *  - CENTER DRILL             - CENTER DRILL
48 LBL 47                        - PECKING
49 L X+0 Y+1 R0 F9998            - TAPPING
50 L Z+08                      END PGM FGLIEDER
51 L Z+09 R0 F07
52 L X+5 Y+10 R0 F06
53 L Z+08 R0 F9998
54 FN 1: Q20 = +Q20 + +8.8
55 FN 9: IF +1 EQU +1 GOTO LBL 111
56 *  - PECKING
```

FL FLT FC FCT FPOL BLK FORM INSERT SECTION CHANGE WINDOW

To insert a structuring block in the (left) program window

▶ Select the block after which the structuring block is to be inserted.

| INSERT SECTION | ▶ Press the INSERT SECTION soft key. |

▶ Enter the structuring text with the alphabetic keyboard.

If necessary, change the level with the CHANGE LEVEL soft key.

To insert a structuring block in the (right) structure window

▶ Select the structuring block after which the new block is to be inserted.

▶ Enter the text with the alphabetic keyboard — the TNC automatically inserts the new block.

Selecting blocks in the program structure window

If you are scrolling through the program structure window block by block, the TNC at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

4.6 Adding Comments

You can add comments to any desired block in the part program to explain program steps or make general notes. There are three possibilities to add comments:

1. To enter comments during programming:

▶ Enter the data for a program block, then press the semicolon key ";" on the alphabetic keyboard — the TNC displays the dialog prompt
COMMENT ?

▶ Enter your comment and conclude the block by pressing the END key.

2. To insert comments after program entry:

▶ Select the block to which a comment is to be added.

▶ Select the last word in the block with the right arrow key:
A semicolon appears at the end of the block and the TNC displays the dialog prompt COMMENT ?

▶ Enter your comment and conclude the block by pressing the END key.

3. To enter a comment in a separate block:

▶ Select the block after which the comment is to be inserted.

▶ Initiate the programming dialog with the semicolon key ";" on the alphabetic keyboard.

▶ Enter your comment and conclude the block by pressing the END key.

```
EDIT        PROGRAMMING AND EDITING
TOOL TABLE    COMMENT?
6   L X+50 Y+50 R0 F MAX M8
7   L Z-5 R0 F MAX
8   CC X+0 Y+0
9   LP PR+14 PA+45 RR F500
10   RND R1 ; APPROACH█
11   FC DR+ R2,5 CLSD+
12   FLT AN+180,925
13   FCT DR+ R10,5 CCX+0 CCY+0
14   FSELECT 1
15   FLT AN+269,025
16   RND R2,5
17   FL AN+0,975
18   FCT DR+ R10,5 CCX+0 CCY+0
19   FLT AN+89,025
20   FCT DR+ R2,5 CLSD-
```

4.7 Creating Text Files

You can use the TNC's text editor to write and edit texts.

Typical applications:

- Recording test results
- Documenting working procedures
- Keeping formulas and creating cutting data diagrams

Text files are type .A files (ASCII files). If you want to edit other types of files with the text editor, you must first convert them into type .A files (see page 89).

To create and leave a text file:

▶ Select the PROGRAMMING AND EDITING mode of operation.

▶ To call the file manager, press the PGM MGT soft key.

▶ To display type .A files, press the SELECT TYPE and then the SHOW .A soft keys.

▶ Select a file and open it with the SELECT soft key or ENT key, **or** create a new file by entering the new file name and confirming your entry with the ENT key.

To leave the text editor, call the file manager and select a file of a different file type, for example a part program.

Editing texts
The first line of the text editor is an information headline which displays the file name, and the location and writing mode of the cursor:

FILE: Name of the text file

LINE: Line in which the cursor is presently located

COLUMN: Column in which the cursor is presently located

INSERT: Insert new text, pushing the existing text to the right

OVERWRITE: Write over the existing text, erasing it where it is replaced with the new text.

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

The line in which the cursor is presently located is depicted in a different color. A line can have up to 77 characters. To start a new line, press the RET key.
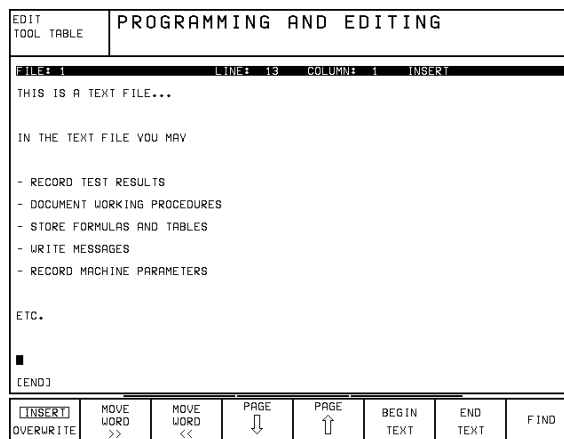
Cursor movements	Soft key
Move one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to the next screen page	PAGE ⇩
Go to the previous screen page	PAGE ⇧
Go to beginning of file	BEGIN TEXT
Go to end of file	END TEXT

Editing functions	Key
Begin a new line	RET
Erase character to left of cursor (backspace)	X
Insert a blank space	SPACE

Erasing and inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text. See the table at right.

To move a word or line to a different location:

▶ Move the cursor to the word or line you wish to erase and insert at a different place in the text.

▶ Press the DELETE WORD or DELETE LINE soft key: The text is deleted and stored temporarily.

▶ Move the cursor to the location where you wish insert the text, and press the RESTORE LINE/WORD soft key.

Delete functions	Soft key
Delete and temporarily store a line	DELETE LINE
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a character	DELETE CHAR
Insert a line or word from temporary storage	RESTORE LINE/WORD

Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before carrying out any of these editing functions, you must first select the desired text block:

▶ To select a text block, move the cursor to the first character of the text you wish to select.

SELECT BLOCK ▶ Press the SELECT BLOCK soft key.

▶ Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys — the selected text is shown in a different color.

```
MANUAL          PROGRAMMING AND EDITING
OPERATION

FILE: 3516              LINE: 10    COLUMN: 27  INSERT
0  BEGIN PGM 3516 MM
1  BLK FORM 0.1 Z X-90 Y-90 Z-40
2  BLK FORM 0.2 X+90 Y+90 Z+0
3  TOOL DEF 50
4  TOOL CALL 1 Z S1400
5  L Z+50 R0 F MAX
6  L X+0 Y+100 R0 F MAX M3
7  L Z-20 R0 F MAX
8  L X+0 Y+80 RL F250
9  FPOL X+0 Y+0
10 FC DR- R80 CCX+0 CCV+0█
11 FCT DR- R7.5
12 FCT DR+ R90 CCX+69.282 CCV-40
13 FSELECT 2
```

| SELECT BLOCK | REMOVE BLOCK | INSERT BLOCK | REMOVE/ INSERT BLOCK | | | APPEND TO FILE | READ FILE |

After selecting the desired text block, you can edit the text with the following soft keys:

Function	Soft key
Delete the selected text and store temporarily	REMOVE BLOCK
Store marked block temporarily without erasing (copy)	REMOVE/ INSERT BLOCK

If necessary, you can now insert the temporarily stored block at a different location:

▶ Move the cursor to the location where you want to insert the temporarily stored text block.

INSERT BLOCK ▶ Press the INSERT BLOCK soft key — the text block is inserted.

You can insert the temporarily stored text block as often as desired.

To transfer the selected text to a different file:

▶ Select the text block as described previously.

| APPEND TO FILE | ▶ Press the APPEND TO FILE soft key — the TNC displays the dialog prompt DESTINATION FILE = |

    ▶ Enter the path and name of the target file. The TNC appends the selected text to the end of the specified file. If no target file with the specified name is found, the TNC creates a new file with the selected text.

To insert another file at the cursor position:

▶ Move the cursor to the location in the text where you wish to insert another file.

| READ FILE | ▶ Press the soft key — the TNC displays the dialog prompt FILE NAME = |

    ▶ Enter the path and name of the file you want to insert.

Finding text sections

With the text editor, you can search for words or character strings in a text. Two functions are available:

1. Finding the current text
The search function is to find the next occurrence of the word in which the cursor is presently located:

▶ Move the cursor to the desired word.

▶ To select the search function, press the FIND soft key.

▶ Press the FIND CURRENT WORD soft key.

2. Finding any text

▶ To select the search function, press the FIND soft key. The TNC displays the dialog prompt FIND TEXT:

▶ Enter the text that you wish to find.

▶ To find the text, press the EXECUTE soft key.

To leave the search function, press the END soft key.

```
MANUAL          PROGRAMMING AND EDITING
OPERATION       FIND TEXT : L Z+100

FILE: 3516          LINE:  0      COLUMN:  1    INSERT

0   BEGIN PGM 3516 MM
1   BLK FORM 0.1 Z X-90 Y-90 Z-40
2   BLK FORM 0.2 X+90 Y+90 Z+0
3   TOOL DEF 50
4   TOOL CALL 1 Z S1400
5   L Z+50 R0 F MAX
6   L X+0 Y+100 R0 F MAX M3
7   L Z-20 R0 F MAX
8   L X+0 Y+80 RL F250
9   FPOL X+0 Y+0
10  FC DR- R80 CCX+0 CCY+0
11  FCT DR- R7.5
12  FCT DR+ R90 CCX+69.282 CCY-40
13  FSELECT 2

 FIND
CURRENT                          EXECUTE   END
 WORD
```

4.8 Integrated Pocket Calculator

The TNC features an integrated pocket calculator with the basic mathematical functions.

With the CALC key you can open and close an additional window for calculations. You can move the window to any desired location on the TNC screen with the arrow keys.

The calculator is operated with short commands through the alphabetic keyboard. The commands are shown in a special color in the calculator window:

```
MANUAL          PROGRAMMING AND EDITING
OPERATION       MISCELLANEOUS FUNCTION M ?

3  TOOL CALL 1 Z S3150
4  L Z+250 R0 F MAX
5  L X-20 Y+50 R0 F MAX
6  L Z-5 R0 F2000
   L X+10 Y+5 RL F100 M3
7  END PGM NEU MM

                              0
                              ARC SIN COS TAN  7 8 9
                              +  -  *  :   4 5 6
                              X^Y SQR 1/X PI  1 2 3
                              ( ) CE  =   0 . +/-
```

Mathematical function	Command
Addition	+
Subtraction	–
Multiplication	*
Division	:
Sine	S
Cosine	C
Tangent	T
Arc sine	AS
Arc cosine	AC
Arc tangent	AT
Powers	^
Square root	Q
Inversion	/
Parenthetic calculations	( )
π (3.14159265359)	P
Display result	=

If you are writing a program and the programming dialog is active, you can use the actual-position-capture key to transfer the result to the highlight position in the current block.
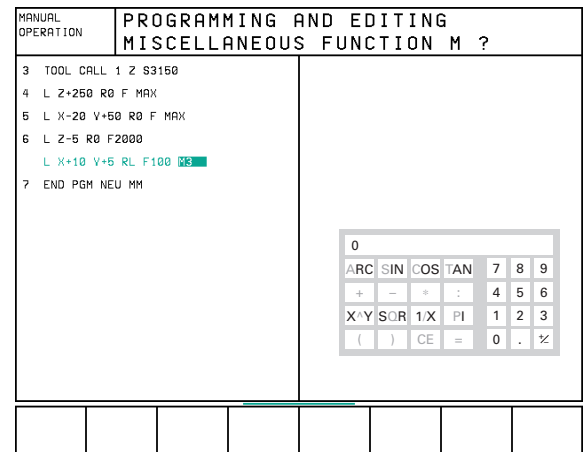
4.9 Creating Pallet Tables

> Pallet tables are managed and output as determined in the PLC. The machine tool manual provides further information.

Pallet tables are used for machining centers with pallet changers: The pallet table calls the part programs that are required for the different pallets, and activates the associated datum tables.

Pallet tables contain the following information:

■ Pallet number PAL

■ Part program name PGM NAME

■ Datum table DATUM

To select a pallet table:

▶ In the PROGRAMMING AND EDITING mode of operation, select the file manager with the PGM MGT key.

▶ To show .P type pallet files, press the SELECT TYPE and SHOW .P soft keys.

▶ Select a pallet table with the arrow keys, or enter a new file name to create a new table.

▶ Confirm you entry with the ENT key.

▶ To link programs and datum tables in a pallet table, enter the program names and the associated datum tables in the corresponding columns of the pallet table. You can move the highlight with the arrow keys within the pallet table. When you are editing a pallet file, the TNC displays the edit soft keys (see table at right).

To leave the pallet file:

▶ To select the file manager, press the Taste PGM MGT key.

▶ To select a different type of file, press the SELECT TYPE soft key and the soft key for the desired file type, for example SHOW .H.

▶ Choose the desired program.

```
MANUAL        PROGRAM TABLE EDITING
OPERATION     PROGRAM NAME ?

   FILE: NEU
PAL  PROGRAM           DATUM
0
1    TNC:\GEHAEUSE\DEC1.H  TNC:\DATUM\DEC1.D
2    TNC:\GEHAEUSE\DEC2.H  TNC:\DATUM\DEC2.D
3    TNC:\A35001\PLATTE.H  TNC:\DATUM\PLATTE.D
4    TNC:\3DPARTS\ZVL35.H  TNC:\DATUM\ZVL35.D
5
6    TNC:\ISOPGM\SURF1.H   TNC:\DATUM\SURF1.D
7
8
9
10
[END]
```

| BEGIN TABLE | END TABLE | PAGE ⇩ | PAGE ⇧ | INSERT LINE | DELETE LINE | NEXT LINE | |

Function	Soft key
Go to the beginning of the table	BEGIN TABLE
Go to the end of the table	END TABLE
Go to the next page of the table	PAGE ⇩
Go to the previous page of the table	PAGE ⇧
Insert the last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to the beginning of the next line	NEXT LINE

5

Programming:

Tools

5.1 Entering Tool-Related Data

Feed rate F

The feed rate is the speed (in millimeters per minute or inches per minute) at which the tool center moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.

Input
You can enter the feed rate in every positioning block. For further information refer to seciton 6.2 "Fundamentals of Path Contours."

Rapid traverse
If you wish to program rapid traverse, enter F MAX with the ENT key as soon as the dialog question "FEED RATE F = ?" appears on the TNC screen.

Duration of effect
A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. F MAX is only effective in the block in which it is programmed. After the block with F MAX is executed, the feed rate will return to the last feed rate entered as a numerical value.

Changing the feed rate during program run
You can adjust the feed rate during program run with the feed-rate override knob.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a TOOL CALL block.

Programmed change
In the part program, you can change the spindle speed in a TOOL CALL block by entering the spindle speed only:

▶ To program a tool call, press the TOOL CALL key.

▶ Ignore the dialog question for "TOOL NUMBER ?" with the NO ENT key.

▶ Ignore the dialog question for "WORKING SPINDLE AXIS X/Y/Z ?" with the NO ENT key.

▶ Enter the new spindle speed for the dialog question "SPINDLE SPEED S= ?".

Changing the spindle speed during program run
You can adjust the spindle speed during program run with the spindle-speed override knob.
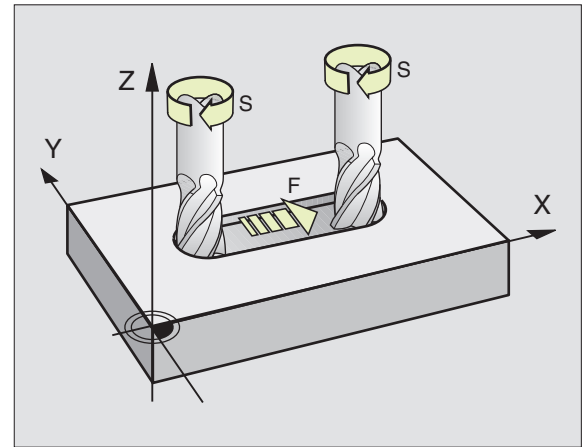
5.2 Tool Data

You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the TNC to calculate the tool center path — i.e. the tool compensation — you must also enter the length and radius of each tool you are using.

Tool data can be entered either directly in the part program with TOOL DEF or separately in a tool table. In a tool table, you can also enter additional data on the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

Tool numbers and tool names
Each tool is identified by a number between 0 and 254. If you are working with tool tables, you can also enter a tool name for each tool.

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool 0 should also be defined with L=0 and R=0.

Tool length L
There are two ways to determine the tool length L:

1 The length L is the difference between the length of the tool and that of a zero tool L_0.

For the algebraic sign:

■ The tool is longer than the zero tool: $L > L_0$

■ The tool is shorter than the zero tool: $L < L_0$

To determine the length:

▶ Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).

▶ Set the datum in the tool axis to 0 (datum setting).

▶ Insert the desired tool.

▶ Move the tool to the same reference position as the zero tool.

▶ The TNC displays the difference between the current tool and the zero tool.

▶ Enter the value in the TOOL DEF block or in the tool table by pressing the actual-position-capture key.

2 Determine the tool length L with a tool presetter. This allows you to enter the determined value directly in the TOOL DEF block without further calculations.
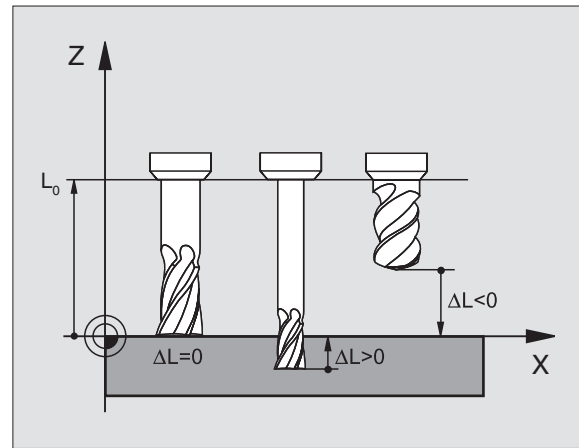
Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii

Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (DR>0). If you are programming the machining data with an allowance, enter the oversize value in the TOOL CALL block of the part program.

A negative delta value describes a tool undersize (DR<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a TOOL CALL block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.

Entering tool data into the program

The number, length and radius of a specific tool is defined in the TOOL DEF block of the part program.

- ▶ To select tool definition, press the TOOL DEF key.
- ▶ Enter the TOOL NUMBER: Each tool is uniquely identified by its number.
- ▶ To enter the TOOL LENGTH, enter the compensation value for the tool length.
- ▶ Enter the TOOL RADIUS.

> ☞ In the programming dialog, you can transfer the value for tool length directly into the input line with the actual-position-capture key. You only need to make sure that the highlight in the status display is placed on the tool axis.

Resulting NC block:

```
4 TOOL DEF 5 L+10 R+5
```
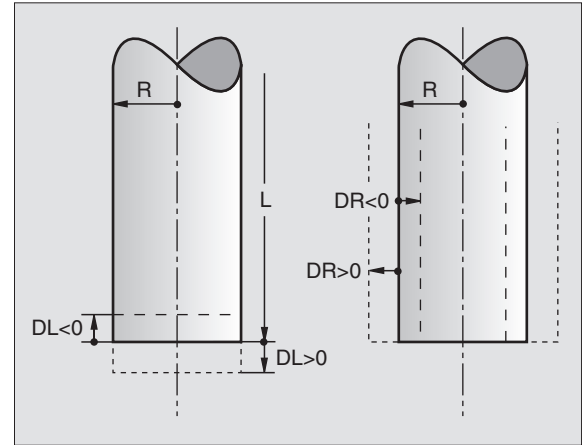
Entering tool data in tables

You can define and store up to 254 tools and their tool data in a tool table. (The maximum number of tools per table can be set in machine parameter 7260).

You must use tool tables if

- your machine tool has an automatic tool changer

- you want to measure tools automatically with the TT 120 touch probe (see section 5.5 "Measuring Tools")

- you want to rough-mill the contour with Cycle 22

(see page 172).

Abbr.	Input	Dialog
T	Number by which the tool is called in the program	–
NAME	Name by which the tool is called in the program	TOOL NAME ?
L	Value for tool length compensation	TOOL LENGTH ?
R	Tool radius R	TOOL RADIUS ?
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	TOOL RADIUS 2 ?
DL	Delta value for tool length	TOOL LENGTH OVERSIZE ?
DR	Delta value for tool radius R	TOOL RADIUS OVERSIZE ?
DR2	Delta value for tool radius R2	TOOL RADIUS OVERSIZE 2 ?
LCUTS	Tooth length of the tool for Cycle 22	TOOTH LENGTH IN TOOL AXIS ?
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycle 22	MAXIMUM PLUNGE ANGLE ?
TL	Tool Lock	TOOL LOCKED? YES = ENT / NO = NO ENT
RT	Number of a replacement tool, if available (see also TIME2)	REPLACEMENT TOOL ?
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.	MAXIMUM TOOL AGE ?
TIME2	Maximum tool life in minutes during TOOL CALL: If the current tool life exceeds this value, the TNC changes the tool during the next TOOL CALL (see also CUR.TIME)	MAXIMUM TOOL AGE FOR TOOL CALL ?
CUR.TIME	Time in minutes the tool has been in use: The TNC automatically counts the current tool life. A starting value can be entered for used tools.	CURRENT TOOL AGE ?
DOC	Comment on tool (up to 16 characters)	TOOL DESCRIPTION ?
PLC	Information on this **tool** that is to be sent to the PLC	PLC STATUS ?

Tool table: Available input data
Tool table: Tool data required for automatic tool measurement

Abbr.	Input	Dialog
CUT.	Number of teeth (20 teeth maximum)	NUMBER OF TEETH ?
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	WEAR TOLERANCE: LENGTH ?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	WEAR TOLERANCE: RADIUS ?
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	CUTTING DIRECTION (M3 = –) ?
TT:R-OFFS	Tool length measurement: tool offset between stylus center and tool center. Preset value: tool radius R	TOOL OFFSET: RADIUS ?
TT:L-OFFS	Tool radius measurement: tool offset in addition to MP6530 (see section 15.1 "General User Parameters") between upper edge of stylus and lower edge of tool. Preset value: 0	TOOL OFFSET: LENGTH ?
LBREAK	Permissible deviation from tool length L for break detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	BREAKAGE TOLERANCE: LENGTH ?
RBREAK	Permissible deviation from tool radius R for break detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	BREAKAGE TOLERANCE: RADIUS ?

Editing tool tables

The tool table that is active during execution of the part program is designated as TOOL.T. You can edit TOOL.T in any of the machine operating modes. Other tool tables that are used for archiving or test runs are given different file names with the extension .T .

To open the tool table TOOL.T:

▶ Select any machine operating mode.

TOOL
TABLE

▶ To select the tool table, press the TOOL TABLE soft key.

EDIT
OFF / ON

▶ Set the EDIT soft key to ON.

To open any other tool table:

▶ Select the PROGRAMMING AND EDITING mode of operation.

PGM
MGT

▶ Call the file manager.

▶ To select the file type, press the SELECT TYPE soft key.

▶ To show type .T files, press the SHOW.T soft key.

▶ Select a file or enter a new file name, and conclude your entry with the ENT key or SELECT soft key.

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys (see figure at upper right). You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table at right.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the symbol >> or << .

To leave the tool table:

▶ Call the file manager and select a file of a different type, e.g. a part program.

When you have opened a tool table for editing, you can enter values at any desired position in the table by moving the cursor with the arrow keys to that position (see figure to the upper right). If all the information in the tool table no longer fits in one screen page, the highlight bar at the top of the table will display the symbol >> or <<.

TABULKA NASTROJU - EDIT				TABULKA PGM EDIT

```
TABULKA NASTROJU - EDIT          TABULKA PGM
RADIUS NASTROJE 2 ?              EDIT

 SOUBR: TOOL              MM                          >>
T   NAME           L          R      R2      DL      DR
0                 -88,4718   +0      +0      +0      +0
1   SCHRUPP_1      +0         +5      +1      +0,05   +0,025
2   SCHRUPP_2      -9         +12,5   +0      +0,05   +0,025
3   SCHRUPP_3      -5,5       +10     +0      +0      +0
4   SCHLICHT_1     -30,2      +4      +4      +0,05   +0,025
5   SCHLICHT_2     -32,59     +16     +16     +0,05   +0,025
6   BOHRER_8       -5,6       +4      +0      +0      +0


AKT.     +X  +250,0000     +Y  +102,3880
         +Z  -114,0914     +C   +30,0000
         +B   +90,0000
T                          F  0          M 5/9
```

BEGIN TABLE	END TABLE	PAGE ⇓	PAGE ⇑		EDIT OFF/ON	NEXT LINE	POCKET TABLE

Editing functions for tool tables	Soft key
Go to the beginning of the table	BEGIN TABLE
Go to the end of the table	END TABLE
Go to the previous table page	PAGE ⇑
Go to the next table page	PAGE ⇓
Go to the beginning of the next line	NEXT LINE
Look for the tool name in the tool table	FIND TOOL NAME
Display / Do not display pocket numbers	SHOW OMIT POCKET NR
Show tool information in columns, or show all information on an individual tool in one screen page	LIST FORMULAR

Additional notes on tool tables

User parameter MP7266 defines which data can be entered in the
tool table and in what sequence the data is displayed.

☞ You can overwrite individual columns or lines of a tool
table with the contents of another file. Prerequisites:

■ The target file must exist.

■ The file to be copied must contain only the columns
(or lines) you want to replace.

To copy individual columns or lines, press the REPLACE
FIELDS soft key.

Pocket table for tool changer

The **TOOL_P** table (for tool pocket) is programmed in a program run
mode of operation.

To select the pocket table:

TOOL TABLE ▶ To call the tool table,
press the TOOL TABLE soft key.

POCKET TABLE ▶ To select the pocket table,
press the POCKET TABLE soft key.

EDIT OFF/ON ▶ Set the EDIT soft key to ON.

You can enter the following information on a tool into a pocket table:

```
TABULKA NASTROJU – EDIT          TABULKA PGM
MISTO BLOKOVAT ANO=ENT/NE=NOENT  EDIT
  SOUBR: TOOL_P
P   T   ST F  L  PLC
0   0             %00000000
1             L   %00000000
2   2       F     %11100011
3             L   %00000000
4   4             %00000000
5   5       F  ■  %00000000
6   6             %00000000

AKT.    ÷X  +250,0000    ÷Y   +102,3880
        ÷Z  -114,0914    ÷C    +30,0000
        ÷B   +90,0000
T                        F 0        M 5/9
```

Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	Tool number	TOOL NUMBER ?
F	Fixed tool number. The tool is always returned to the same pocket.	FIXED POCKET ? YES = ENT / NO = NO ENT
L	Locked pocket	POCKET LOCKED ? YES = ENT / NO = NO ENT
ST	Special tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked (status L).	SPECIAL TOOL ?
PLC	Information on this **tool pocket** that is to be sent to the PLC	PLC STATUS ?

Calling tool data

A TOOL CALL block in the part program is defined with the following data:

TOOL CALL
- Select the tool call function with the TOOL CALL key.
- WORKING SPINDLE AXIS X/Y/Z: Enter the tool axis.
- TOOL NUMBER: Enter the number or name of the tool. The tool must already be defined in a TOOL DEF block or in the tool table. To call a tool by the tool name, enter the name in quotation marks. The tool name always refers to the entry in the active tool table TOOL .T.
- SPINDLE SPEED S
- TOOL LENGTH OVERSIZE: Enter the delta value for the tool length.
- TOOL RADIUS OVERSIZE: Enter the delta value for the tool radius.

Example:
Call tool number 5 in the tool axis Z with a spindle speed 2500 rpm. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius with an undersize of 1 mm.

```
20 TOOL CALL 5 Z S2500 DL+0.2 DR-1
```

The character D preceding L and R designates delta values.

Tool preselection with tool tables
If you are working with tool tables, use TOOL DEF to preselect the next tool. Simply enter the tool number or a corresponding Q parameter, or type the tool name in quotation marks.

Tool change

The tool change function can vary depending on the individual machine tool. You machine manual provides more information on this function.

Tool change position
A tool change position must be approachable without collision. With the miscellaneous functions M91 and M92, you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position. If TOOL CALL 0 is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

Manual tool change
To change the tool manually, stop the spindle and move the tool to the tool change position:

- Move to the tool change position under program control.
- Interrupt program run (see section 11.4 "Program Run").
- Change the tool.
- Resume the program run (see section 11.4 "Program Run").

Automatic tool change
If your machine tool has automatic tool changing capability, the program run is not interrupted. When the TNC reaches a TOOL CALL, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

This function can vary depending on the individual machine tool. Your machine manual provides more information on M101.

The TNC automatically changes the tool if the tool life TIME1 or TIME2 expires during program run. To use this miscellaneous function, activate M101 at the beginning of the program. M101 is reset with M102.

The tool is not always changed immediately, but, depending on the workload of the control, a few NC blocks later.

Prerequisites for standard NC blocks with radius compensation R0, RR, RL
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

Prerequisites for NC blocks with surface-normal vectors and 3-D compensation
The radius of the replacement tool can differ from the radius of the original tool. The tool radius is not included in program blocks transmitted from CAD systems. You can enter a negative delta value (DR) in the tool table.

If DR is positive, the TNC displays an error message and does not replace the tool. You can suppress this message with the M function M107, and reactivate it with M108.

5.3 Tool Compensation

The TNC adjusts the spindle path in the tool axis by the compensation value for the tool length. In the working plane, it compensates the tool radius.

If you are writing the part program directly on the TNC, the tool radius compensation is effective only in the working plane. The TNC accounts for the compensation value in up to five axes including the rotary axes.

☞ If a part program generated by a CAD system contains surface-normal vectors, the TNC can perform three-dimensional tool compensation. For more information, refer to section 5.4 "Three-Dimensional Tool Compensation."

Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called and the tool axis moves. To cancel length compensation call a tool with the length L=0.

☞ If you cancel a positive length compensation with TOOL CALL 0, the distance between tool and workpiece will be reduced.

After TOOL CALL, the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation, the TNC takes the delta values from both the TOOL CALL block and the tool table into account:

Compensation value = $L + DL_{TOOL\ CALL} + DL_{TAB}$, where

L	is the tool length L from the TOOL DEF block or tool table
$DL_{TOOL\ CALL}$	is the oversize for length DL in the TOOL CALL block (not taken into account by the position display)
DL_{TAB}	is the oversize for length DL in the tool table

Tool radius compensation

The NC block for programming a tool movement contains:

- RL or RR for compensation in the tool radius
- R+ or R– for radius compensation in single-axis movements
- R0 if no radius compensation is required

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with RL or RR. To cancel radius compensation, program a positioning block with R0.
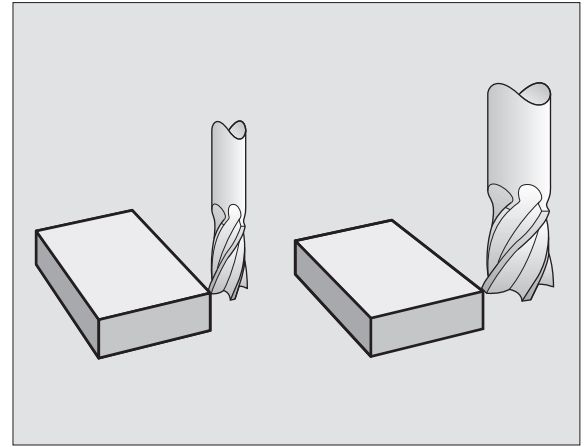
For tool radius compensation, the TNC takes the delta values from both the TOOL CALL block and the tool table into account:

Compensation value = R + $DR_{TOOL\ CALL}$ + DR_{TAB} , where

R	is the tool radius R from the TOOL DEF block or tool table
$DR_{TOOL\ CALL}$	is the oversize for radius DR in the TOOL CALL block (not taken into account by the position display)
DR_{TAB}	is the oversize for radius DR in the tool table

Tool movements without radius compensation: R0
The tool center moves in the working plane to the programmed path or coordinates.

Applications: Drilling and boring, pre-positioning
(see figure at right)

Tool movements with radius compensation: RR and RL

RR The tool moves to the right of the programmed contour

RL The tool moves to the left of the programmed contour

The tool center moves along the contour at a distance equal to the radius. "Right" or "left" are to be understood as based on the direction of tool movement along the workpiece contour (see illustrations on the next page).

☞ Between two program blocks with different radius compensations (RR and RL) you must program at least one block without radius compensation (that is, with R0).

Radius compensation does not come into effect until the end of the block in which it is first programmed.

Whenever radius compensation is activated with RR/RL or canceled with R0, the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.
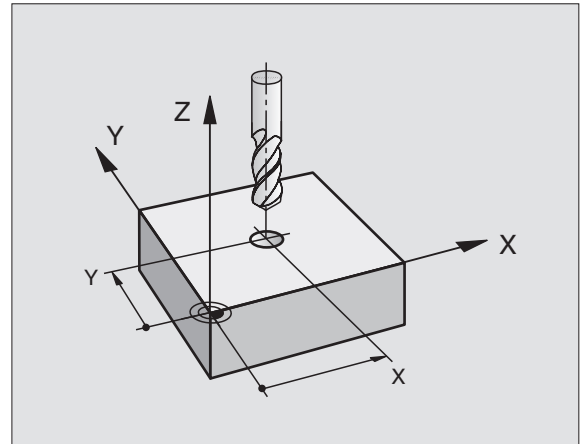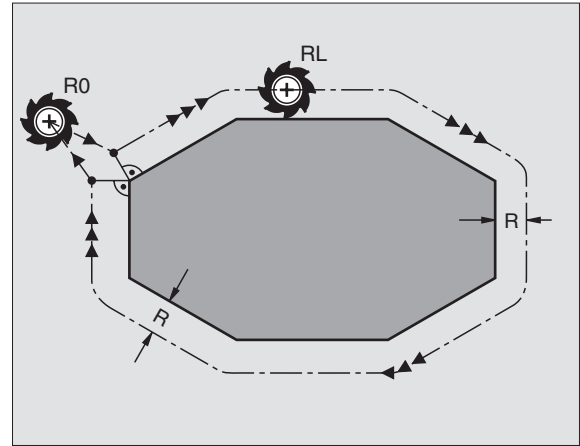
Entering radius compensation

When you program a path contour, the following dialog question is displayed after entry of the coordinates:

RADIUS COMP. RL/RR/NO COMP. ?

| RL | To select tool movement to the left of the contour, press the RL soft key, or |

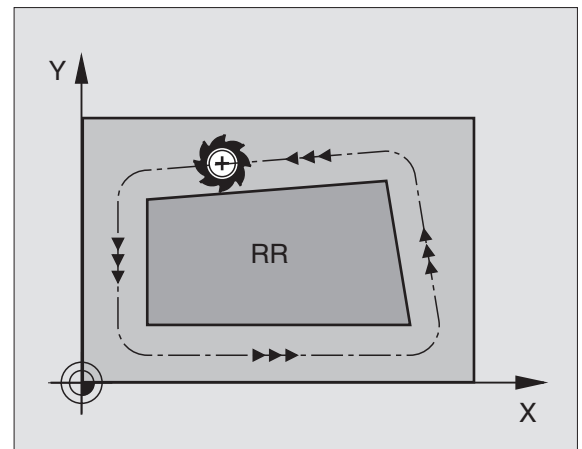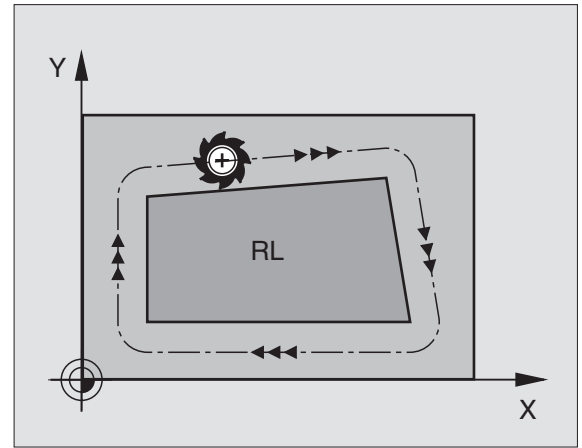| RR | To select tool movement to the right of the contour, press the RR soft key, or |

| ENT | To select tool movement without radius compensation or to cancel radius compensation, press the ENT key. |

| END | To terminate the dialog, press the END key. |

Radius compensation: Machining corners

Outside corners
If you program radius compensation, the TNC moves the tool in a transitional arc around corners. The tool "rolls around" the corner point. If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.

Inside corners
The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.

☞ To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.

Machining corners without radius compensation
If you program the tool movement without radius compensation, you can change the tool path and feed rate at workpiece corners with the miscellaneous functions M90 and M112 (see section 7.4 "Miscellaneous Functions for Contouring Behavior").
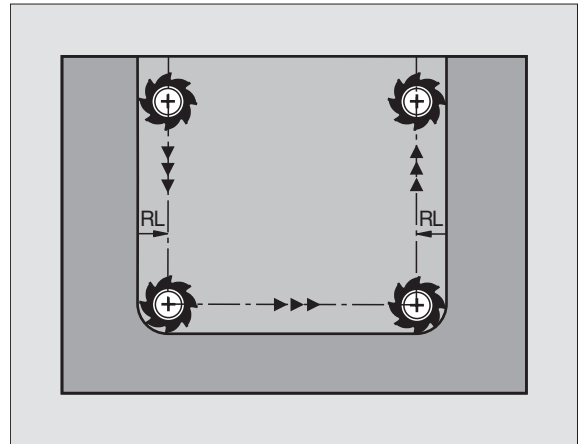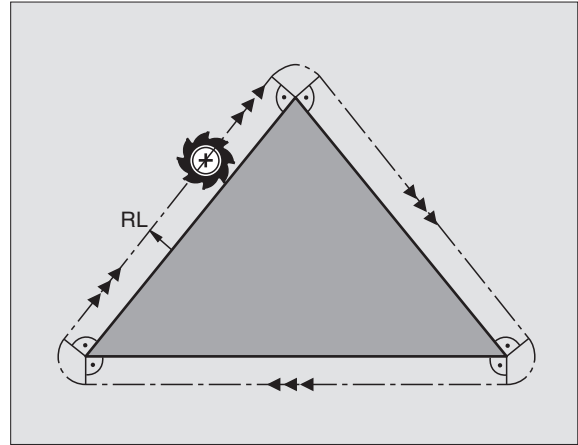
5.4 Three-Dimensional Tool Compensation

The TNC can carry out a three-dimensional tool compensation (3-D compensation) for straight-line blocks. Apart from the X, Y and Z coordinates of the straight-line end point, these blocks must also contain the components NX, NY and NZ of the surface-normal vector (see definition below). The straight-line end point and the surface normal vector are calculated by a CAD system. The 3-D compensation enables you to use tools that have other dimensions than the ones you originally programmed.

Tool shapes

The available tool shapes (see the upper two illustrations at right) are defined by the tool radii R and R2:

TOOL RADIUS: R
Distance from the tool center to the tool circumference

TOOL RADIUS 2: R2
Radius of the curvature between tool tip and tool circumference.

The ratio of R to R2 determines the shape of the tool:

R2 = 0   End mill

R2 = R   Toroid cutter

0 < R2 < R   Spherical cutter

These data also specify the coordinates of the tool datum P_T.

You enter the values for TOOL RADIUS and TOOL RADIUS 2 in the tool table.

Surface-normal vectors

Definition of surface-normal vectors
A surface-normal vector is a mathematical quantity with

- a magnitude
  here: the distance between the workpiece surface and the tool datum P_T , and

- a direction
  here: perpendicular to the workpiece surface to be machined, and pointing away from the workpiece toward the tool datum P_T .

The magnitude and direction of a surface-normal vector is determined by the components NX, NY and NZ.
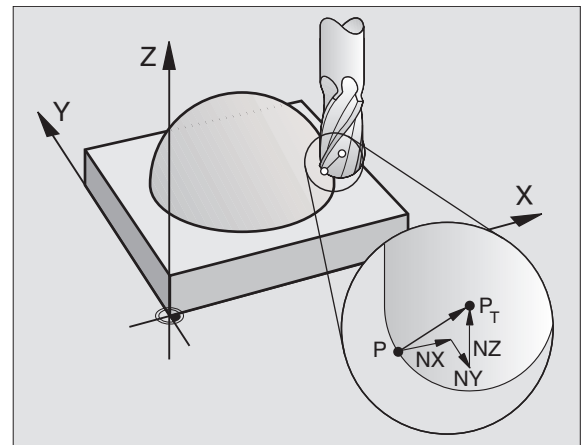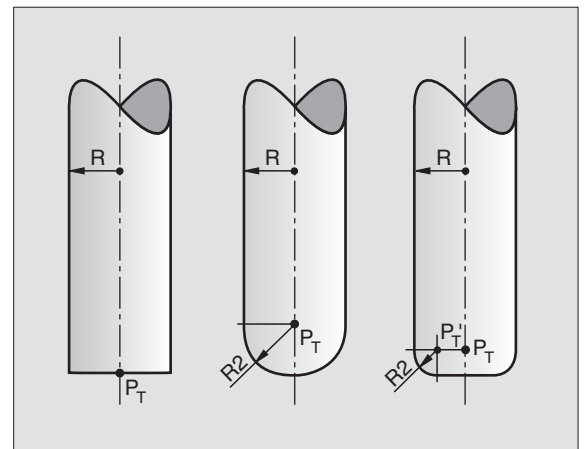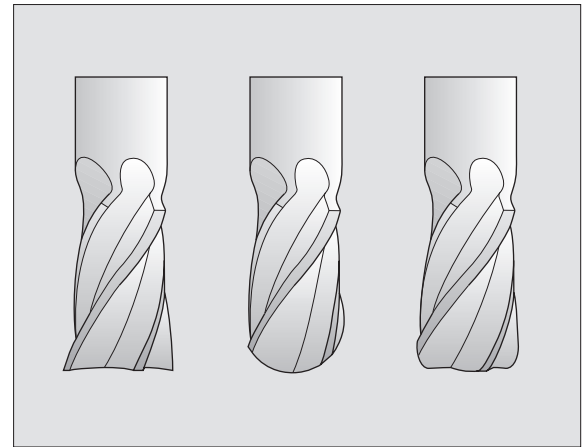
☞ The coordinates for the X, Y, Z positions and the surface-normal components NX, XY, XZ must be in the same sequence in the NC block.

3-D compensation with surface-normal vectors is only effective for coordinates in the main axes X, Y, Z.

The TNC will **not** display an error message if an entered tool oversize would cause damage to the contour.

Machine parameter 7680 defines whether the CAD system calculates the tool length compensation from the center of sphere P_T or the south pole of the sphere P_{SP}.
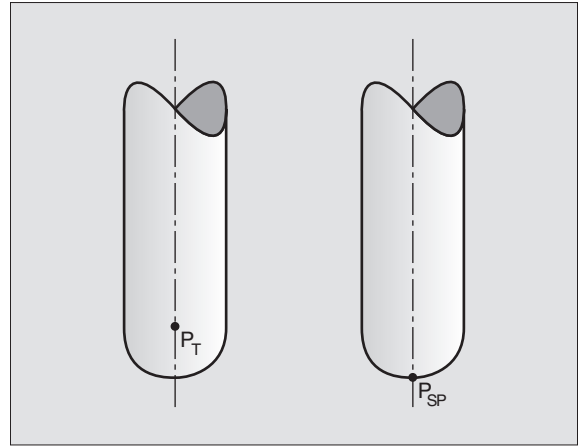
Compensating other tool dimensions: Delta values

If you want to use tools that have different dimensions than the ones you originally programmed, you can enter the difference between the tool lengths and radii in the tool table as delta values:

■ Positive delta value DL, DR, DR2
The tool is larger than the original tool (oversize).

■ Negative delta value DL, DR, DR2
The tool is smaller than the original tool (undersize).

The TNC corrects the tool position by the delta values and the surface-normal vector.

Example: Program block with a surface-normal vector

LN	X+31.737 Y+21.954 Z+33.165 NX+0.2637581
	NY+0.0078922 NZ−0.8764339 F1000 M3

LN   Straight line with 3-D compensation

X, Y, Z   Compensated coordinates of the straight-line end point

NX, NY, NZ   Components of the surface-normal vector

F   Feed rate

M   Miscellaneous function

The feed rate F and miscellaneous function M can be entered and changed in the PROGRAMMING AND EDITING mode of operation. The coordinates of the straight-line end point and the components of the surface-normal vector are defined by the CAD system.
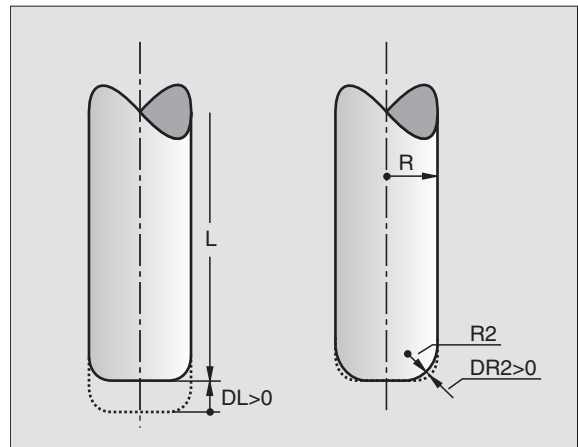
5.5 Measuring Tools with the TT 120 Touch Probe

> The TNC and the machine tool must be set up by the machine tool builder for use of the TT 120 touch probe.
>
> Some cycles and functions may not be provided on your machine tool. Refer to your machine manual for more information on the available cycles and functions.

In conjunction with the TNC's tool measurement cycles, the TT 120 enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for in the next tool call. The following types of tool measurement are provided:

■ Tool measurement while the tool is at standstill

■ Tool measurement while the tool is rotating

■ Measurement of individual teeth

You can define the tool measurement cycles in the PROGRAMMING AND EDITING mode of operation. The following cycles are available:

■ TCH PROBE 30.0 TT CALIBRATE

■ TCH PROBE 31.0 TOOL LENGTH

■ TCH PROBE 32.0 TOOL RADIUS

> The measuring cycles can be used only when the central tool file TOOL.T is active.
>
> Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with TOOL CALL.
>
> You can also measure tools in a tilted working plane.

Setting the machine parameters

> The TNC uses the feed rate for probing defined in MP6520 when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$$n = \frac{MP6570}{r \bullet 0.0063}$$

where

n	is the spindle speed in rpm
MP6570	is the maximum permissible cutting speed in m/min
r	is the active tool radius in mm

The feed rate for probing is calculated from:

v = meas. tolerance $\bullet$ n

where

v	is the feed rate for probing in mm/min
meas. tolerance	is the measuring tolerance in mm, depending on MP6507
n	is the speed in 1/min

MP6507 determines the calculation of the probing feed rate:

MP6507=0:
The measuring tolerance remains constant — regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (MP6570) and the permissible tolerance (MP6510), the sooner you will encounter this effect.

MP6507=1:
The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Meas. tolerance
up to 30 mm	MP6510
30 to 60 mm	2 $\bullet$ MP6510
60 to 90 mm	3 $\bullet$ MP6510
90 to 120 mm	4 $\bullet$ MP6510

MP6507=2:
The feed rate for probing remains constant, the error of measurement, however, rises linearly with the increase in tool radius:

$$\text{Meas. tolerance} = \frac{r \bullet MP6510}{5\,mm}$$

where

r	is the tool radius in mm
MP6510	is the maximum permissible error of measurement

Displaying the measuring results

You can display the results of tool measurement in the additional status display by pressing the STATUS TOOL PROBE soft key (in the machine operating modes). The TNC then shows the program blocks in the left and the measuring results in the right screen window. The measuring results that exceed the permissible wear tolerance are marked in the status display with an asterisk "∗"; the results that exceed the permissible breakage tolerance are marked with the character B.

Calibrating the TT 120 touch probe

☞ Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT 120 within the machine working space must be defined by setting the machine parameters 6580.0 to 6580.2.

If you change the setting of any of the machine parameters 6580.0 to 6580.2, you must recalibrate the TT 120.

The TT 120 is calibrated automatically with the measuring cycle TCH PROBE 30. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle. The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.

TOUCH PROBE
▶ To program the calibration cycle, press the TOUCH PROBE key in the PROGRAMMING AND EDITING mode of operation.

▶ TCH PROBE 30 TT CALIBRATE: Select the measuring cycle 30 TT CALIBRATE with the arrow keys and confirm with ENT.

▶ CLEARANCE HEIGHT: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures.

PROGRAM/PROVOZ PLYNULE   PROGRAM TEST

```
0  BEGIN PGM TT MM
1  TOOL CALL 1 Z
2  TCH PROBE 30.0 TT KALIBROVANI
3  TCH PROBE 30.1 VYSKA: +250
4  TCH PROBE 31.0 DELKA NASTROJE
5  TCH PROBE 31.1 KONTROLA: 0
6  TCH PROBE 31.2 VYSKA: +250
7  TCH PROBE 31.3 PROMERENI BRITU : 1
8  TCH PROBE 32.0 RADIUS NASTROJE
```

NASTROJ

R   MIN 2   +8.4171
      MAX 1   +8.7554
      DVN   +8.8964

```
1    +8.7554 *
2    +8.4171 *
3    +8.7293 *
4    +8.7464 *
```

AKT.   +X   +250,0000   +Y   +102,3880
       +Z   −114,0914   +C   +30,0000
       +B   +90,0000

T   F 0   M 5/9

| PAGE ⇧ | PAGE ⇩ | BEGIN TEXT | END TEXT | RESTORE POS. AT N | | /▢ OFF/ON |

Resulting NC blocks

```
6 TOOL CALL 1 Z
7 TCH PROBE 30.0 CALIBRATE TT
8 TCH PROBE 30.1 HEIGHT: +90
```

Measuring the tool length

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The tool length is measured with the cycle TCH PROBE 31 TOOL LENGTH. Depending on the input parameters you can measure the length of a tool by one of the following methods:

■ If the tool diameter is larger than the diameter of the measuring surface of the TT 120, you can measure the tool while it is rotating.

■ If the tool diameter is smaller than the diameter of the measuring surface of the TT 120, or if you are measuring the length of a drill or spherical cutter, you can measure the tool while it is at standstill.

■ If the tool diameter is larger than the diameter of the measuring surface of the TT 120, you can measure the individual teeth of the tool while it is at standstill.

Measuring cycle for measuring a tool during rotation
The TNC determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe system and then moving it toward the measuring surface until it contacts the surface. The offset is programmed in the tool table under TOOL OFFSET: RADIUS (TT: R-OFFS).

Measuring cycle for measuring a tool during standstill (e.g. for drills)
The TNC positions the tool to be measured over the center of the measuring surface. It then moves the tool toward the measuring surface of the TT 120 (without rotation of the spindle) until the tool contacts the surface. To activate this function, enter zero for the TOOL OFFSET: RADIUS (TT: R-OFFS) in the tool table.

Measuring cycle for measuring individual teeth
The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in MP6530. You can enter an additional offset with TOOL OFFSET: LENGTH (TT: L-OFFS) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for CUTTER MEASUREMENT.

TOUCH PROBE

▶ To program the measuring cycle, press the TOUCH PROBE key in the PROGRAMMING AND EDITING mode of operation.

▶ TCH PROBE 31 TT TOOL LENGTH: select measuring cycle 31 TT TOOL LENGTH with the arrow keys and confirm with ENT.

▶ MEASURE TOOL=0 / CHECK TOOL=1: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

▶ CLEARANCE HEIGHT: Position in the tool axis in which there is no danger of collision with the workpiece or fixtures.

▶ CUTTER MEASUREMENT 0=NO / 1=YES: Choose whether or not the TNC is to measure the individual teeth.

Resulting NC blocks for measuring a rotating tool for the first time

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 0
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0
```

Resulting NC blocks for inspecting a tool and measuring the individual teeth

```
6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. TOOL LENGTH
8 TCH PROBE 31.1 CHECK: 1
9 TCH PROBE 31.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1
```

Measuring the tool radius

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The tool radius is measured with the cycle TCH PROBE 32 TOOL RADIUS. Depending on the input parameters you can measure the radius of a tool by:

■ Measuring the tool while it is rotating

■ Measuring the tool while it is rotating and subsequently measuring the individual teeth.

Measuring cycle
The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in MP6530. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the TNC measures the radius of each tooth with the aid of oriented spindle stops.

▶ To program the measuring cycle, press the TOUCH PROBE key in the PROGRAMMING AND EDITING mode of operation.

▶ TCH PROBE 32 TT TOOL RADIUS: select measuring cycle 32 TT TOOL RADIUS with the arrow keys and confirm with ENT.

▶ MEASURE TOOL=0 / CHECK TOOL=1: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

▶ CLEARANCE HEIGHT: Position in the tool axis in which there is no danger of collision with the workpiece or fixtures.

▶ CUTTER MEASUREMENT 0=NO / 1=YES: Choose whether or not the TNC is to measure the individual teeth.

Resulting NC blocks for measuring a rotating tool for the first time

```
7  TOOL CALL 12 Z
8  TCH PROBE 32.0 CAL. TOOL RADIUS
9  TCH PROBE 32.1 CHECK: 0
10 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 0
```

Resulting NC blocks for inspecting a tool and measuring the individual teeth

```
7  TOOL CALL 12 Z
8  TCH PROBE 32.0 CAL. TOOL RADIUS
9  TCH PROBE 32.1 CHECK: 1
10 TCH PROBE 32.2 HEIGHT: +120
10 TCH PROBE 31.3 PROBING THE TEETH: 1
```

6

Programming:

Programming Contours

6.1 Overview of Tool Movements

Path functions

A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.

Free contour (FK) programming

If a production drawing is not dimensioned for NC and the dimensions given are not sufficient for creating a part program, you can program the workpiece contour with the FK free contour programming and have the TNC calculate the missing data.

With FK programming, you also program tool movements for **straight lines** and **circular arcs.**

Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

- program run, such as a program interruption
- machine functions, such as switching spindle rotation and coolant supply on and off
- contouring behavior of the tool.

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. If you wish to execute a specific program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.

How subprograms and program section repeats are used in programming is described in Chapter 9.

Programming with Q parameters

Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

In addition, parametric programming enables you to measure with the 3-D touch probe during program run.

Programming with Q parameter is described in Chapter 10.
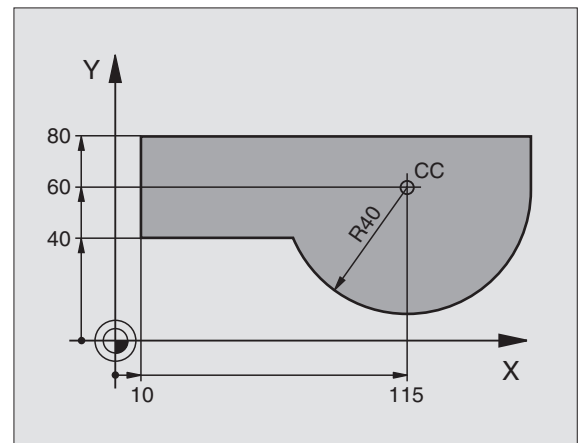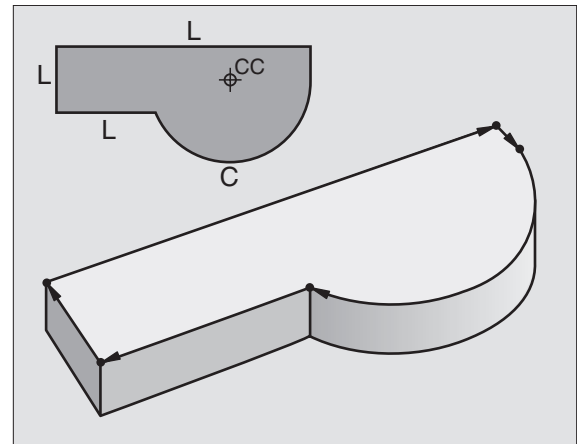
6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

You create a part program by programming the path functions for the individual contour elements in sequence. You usually do this by entering **the coordinates of the end points of the contour elements** given in the production drawing. The TNC calculates the actual path of the tool from these coordinates, and from the tool data and radius compensation.

The TNC moves all axes programmed in a single block simultaneously.

Movement parallel to the machine axes
The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
L  X+100
```

L   Path function for "straight line"

X+100   Coordinate of the end point

The tool retains the Y and Z coordinates and moves to the position X=100 (see figure at upper right).

Movement in the main planes
The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example:

```
L  X+70  Y+50
```

The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see figure at center right).
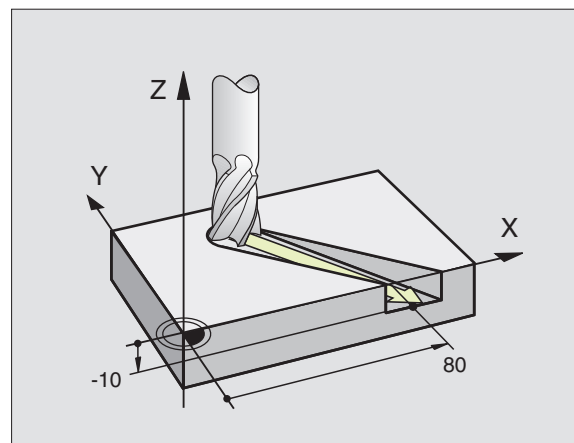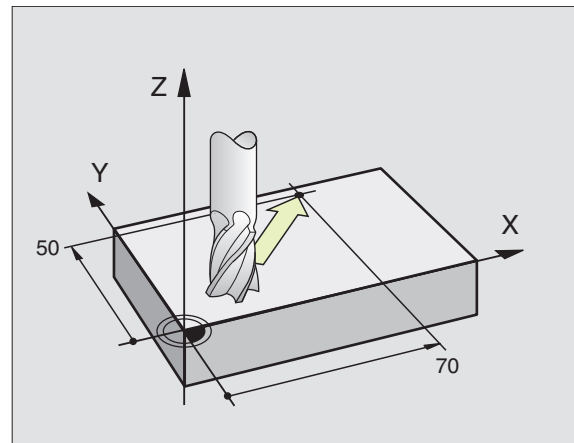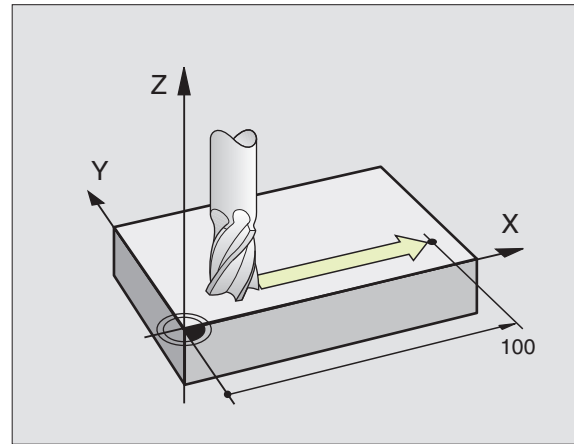
Three-dimensional movement
The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

```
L  X+80  Y+0  Z-10
```

This example is illustrated in the figure at lower right.

Entering more than three coordinates

The TNC can control up to five axes simultaneously (for example, three linear and two rotary axes).

Such programs are too complex to program at the machine, however, and are usually created with a CAD system.

Example:

```
L X+20 Y+10 Z+2 A+15 C+6 R0 F100 M3
```

☞ The TNC graphics cannot simulate movements in more than three axes.

Circles and circular arcs

The TNC moves two axes simultaneously in a circular path relative to the workpiece. You can define a circular movement by entering the circle center CC.

When you program a circle, the TNC assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a tool call:

Spindle axis	Main plane
Z	**XY**, also UV, XV, UY
Y	**ZX**, also WU, ZU, WX
X	**YZ**, also VW, YW, VZ

☞ You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see page 120) or Q parameters (see Chapter 10).

Direction of rotation DR for circular movements

When a circular path has no tangential transition to another contour element, enter the direction of rotation DR:

Clockwise direction of rotation: DR–
Counterclockwise direction of rotation: DR+

Radius compensation

Radius compensation must be programmed before the block containing the coordinates for the first contour element. You cannot begin radius compensation in a circle block — it must be activated beforehand an a straight-line block or approach block (APPR block).

For detailed information on approach and straight-line blocks, refer to sections 6.3 "Contour Approach and Departure" and 6.4 "Path Contours — Cartesian Coordinates."
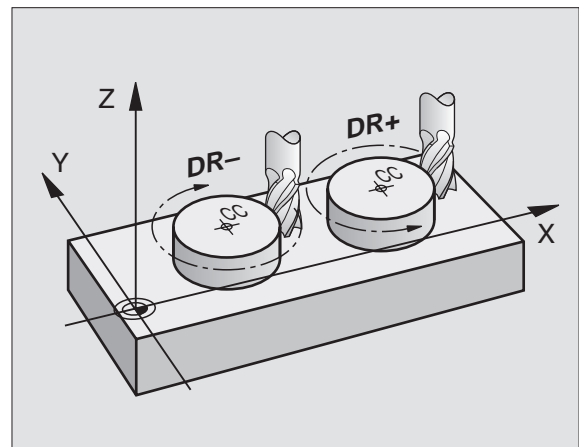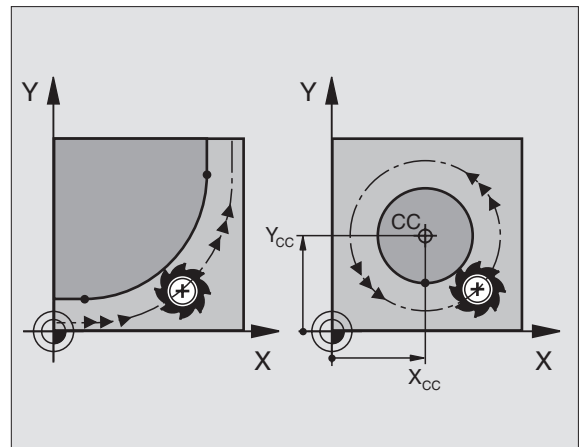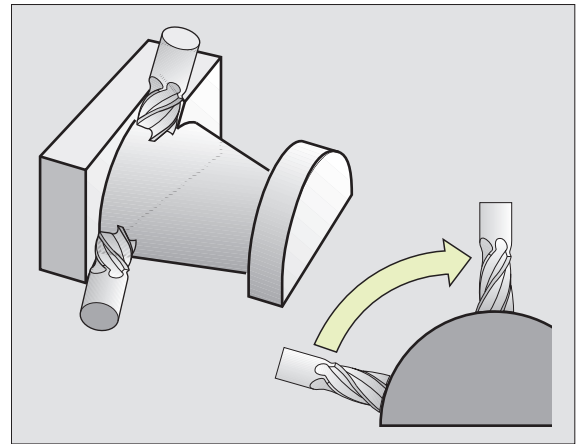
Pre-positioning
Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

Creating the program blocks with the path function keys
The gray path function keys initiate the plain language dialog. The TNC asks you successively for all the necessary information and inserts the program block into the part program.

Example — programming a straight line:

[path function key]	Initiate the programming dialog (here, for a straight line).

COORDINATES ?

X 10	Enter the coordinates of the straight-line end point.
Y 5	
ENT	

TOOL RADIUS COMP. RL/RR/NO COMP. ?

RL	Select the radius compensation (here, press the RL soft key — the tool moves to the left of the programmed contour).

FEED RATE F=? / F MAX = ENT

100 ENT	Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT.

MISCELLANEOUS FUNCTION M ?

3 ENT	Enter a miscellaneous function (here, M3), and terminate the dialog with ENT.

The part program now contains the following line:

`L X+10 Y+5 RL F100 M3`

MANUAL OPERATION

PROGRAMMING AND EDITING
MISCELLANEOUS FUNCTION M ?

```
3  TOOL CALL 1 Z S3150
4  L Z+250 R0 F MAX
5  L X-20 Y+50 R0 F MAX
6  L Z-5 R0 F2000
   L X+10 Y+5 RL F100 M3
7  END PGM NEU MM
```

6.3 Contour Approach and Departure

Overview: Types of paths for contour approach and departure

The functions for contour approach and departure are activated with the APPR/DEP key. You can then select the desired path function with the corresponding soft key:

| Function	Soft keys: Approach	Departure
Straight line with tangential connection	APPR LT	DEP LT
Straight line perpendicular to the contour point	APPR LN	DEP LN
Circular arc with tangential connection	APPR CT	DEP CT
Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside of the contour on a tangentially connecting line.	APPR CT	DEP LCT

Approaching and departing a helix

The tool approaches and departs a helix on its extension by moving in a circular arc that connects tangentially to the contour. You program helix approach and departure with the APPR CT and DEP CT functions.

Important positions for approach and departure

- Starting point P_S
  You program this position in the block before the APPR block. P_S lies outside the contour and is approached without radius compensation (R0).

- Auxiliary point P_H
  Some of the paths for approach and departure go through an auxiliary point P_H that the TNC calculates from your input in the APPR or DEP block.

- First contour point P_A and last contour point P_E
  You program the first contour point P_A in the APPR block. The last contour point P_E can be programmed with any path function.

- If the APPR block also contains a Z axis coordinate, the TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.

- End point P_N
  The position P_N lies outside of the contour and results from your input in the DEP block. If the DEP block also contains a Z axis coordinate, the TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.


```
MANUAL        PROGRAMMING AND EDITING
OPERATION

2  BLK FORM 0.2 X+100 V+100 Z+0
3  TOOL CALL 1 Z S3150
4  L Z+250 R0 F MAX
5  L X-20 V+50 R0 F MAX
6  L Z-5 R0 F2000
7  END PGM NEU MM

APPR LT  APPR LN  APPR CT  APPR LCT  DEP LT  DEP LN  DEP CT  DEP LCT
```
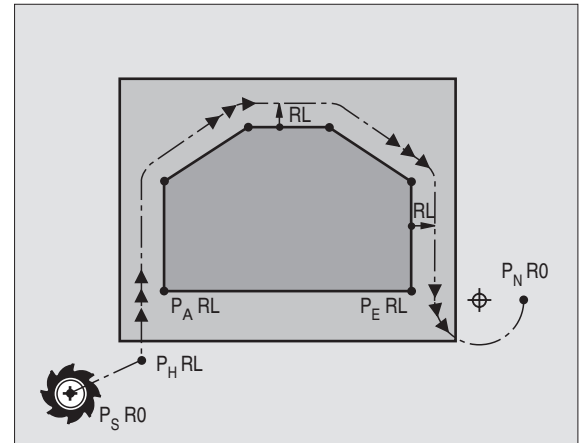
You can enter the position data in absolute or incremental coordinates and in Cartesian or polar coordinates.

The TNC does not check whether the programmed contour will be damaged when moving from the actual position to the auxiliary point P_H. Use the test graphics to simulate approach and departure before executing the part program.

When approaching the contour, allow sufficient distance between the starting point P_S and the first contour point P_A to assure that the TNC will reach the programmed feed rate for machining.

The TNC moves the tool from the actual position to the auxiliary point P_H at the feed rate that was last programmed.

Radius compensation
The tool radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically remove the tool radius compensation.

Contour approach without radius compensation: If you program the APPR block with R0, the TNC will calculate the tool path for a tool radius of 0 mm and a radius compensation RR! The radius compensation is necessary to set the direction of contour approach and departure in the APPR/DEP LN and APPR/DEP CT functions.

## Approaching on a straight line with tangential connection: APPR LT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from P_H to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H is separated from the first contour point P_A by the distance LEN.

▶ Use any path function to approach the starting point P_S.

▶ Initiate the dialog with the APPR/DEP key and APPR LT soft key:

  ▶ COORDINATES of the first contour point P_A

  ▶ LEN: Distance from the auxiliary point P_H to the first contour point P_A

  ▶ RADIUS COMPENSATION for machining

Abbreviation	Meaning
APPR	Approach
DEP	Departure
L	Line
C	Circle
T	Tangential (smooth connection)
N	Normal (perpendicular)
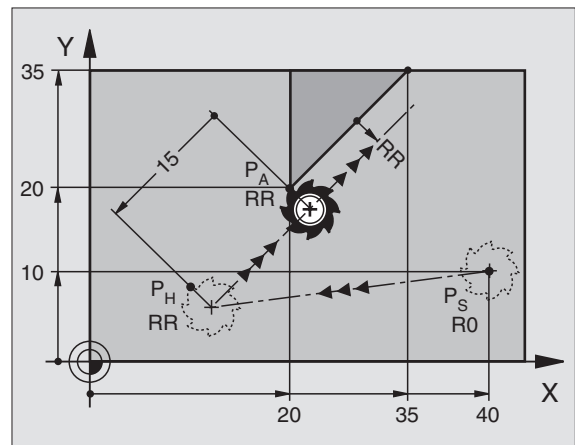
Resulting NC blocks

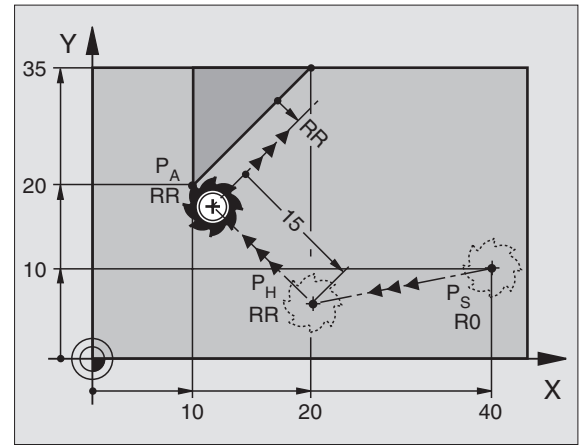|	
7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR, distance P_H to P_A: LEN=15
9 L X+35 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a straight line perpendicular to the first contour point: APPR LN

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from P_H to the first contour point P_A on a straight line that is perpendicular to the first contour element. The auxiliary point P_H is separated from the first contour point P_A by the distance LEN.

▶ Use any path function to approach the starting point P_S.

▶ Initiate the dialog with the APPR/DEP key and APPR LN soft key:

▶ COORDINATES of the first contour point P_A

▶ LENGTH: Distance from the auxiliary point P_H to the first contour point P_A
Always enter LEN as a positive value!

▶ RADIUS COMPENSATION RR/RL for machining

Resulting NC blocks

`7 L X+40 Y+10 R0 FMAX M3`	Approach P_S without radius compensation
`8 APPR LN X+10 Y+20 Z-10 LEN+15 RR F100`	P_A with radius comp. RR, distance P_H to P_A: LEN=15
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

## Approaching on a circular arc with tangential connection: APPR CT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from P_H to the first contour point P_A following a circular arc that is tangential to the first contour element.

The arc from P_H to P_A is determined through the radius R and the center angle CCA. The direction of rotation of the circular arc is automatically derived from the tool path for the first contour element.

▶ Use any path function to approach the starting point P_S.

▶ Initiate the dialog with the APPR/DEP key and APPR CT soft key:

▶ COORDINATES of the first contour point P_A

▶ RADIUS R of the circular arc

■ If the tool should approach the workpiece in the direction defined by the radius compensation:
Enter R as a positive value.

■ If the tool should approach the workpiece opposite to the radius compensation:
Enter R as a negative value.
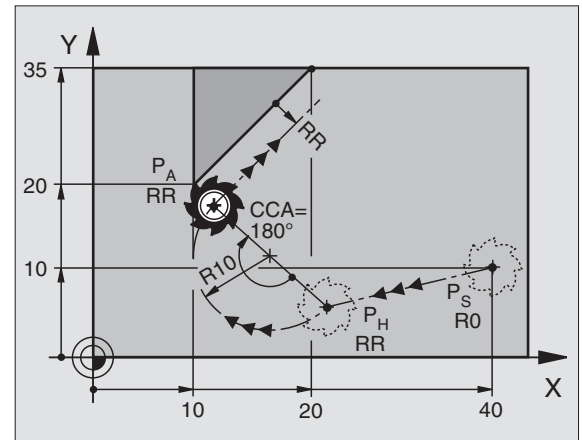
▷ CENTER ANGLE CCA of the arc

■ CCA can only be entered as a positive value.

■ Maximum input value 360°

▷ RADIUS COMPENSATION RR/RL for machining

Resulting NC blocks

`7 L X+40 Y+10 R0 FMAX M3`	Approach P_S without radius compensation
`8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100`	P_A with radius compensation RR, radius R=10
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

## Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from P_H to the first contour point P_A on a circular arc.

The arc is connected tangentially both to the line P_S – P_H as well as to the first contour element. Once these lines are known, the radius then suffices to completely define the tool path.

▷ Use any path function to approach the starting point P_S.

▷ Initiate the dialog with the APPR/DEP key and APPR LCT soft key:

   ▷ COORDINATES of the first contour point P_A

   ▷ RADIUS R of the arc
   Always enter R as a positive value.

   ▷ RADIUS COMPENSATION for machining
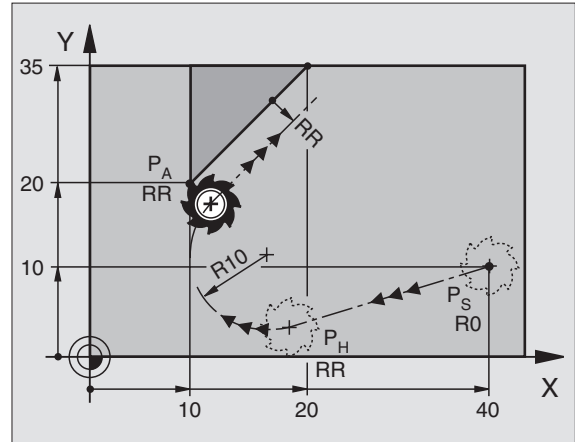
Resulting NC blocks

|	
`7 L X+40 Y+10 R0 FMAX M3`	Approach P_S without radius compensation
`8 APPR LCT X+10 Y+20 Z-10 R10 RR F100`	P_A with radius compensation RR, radius R=10
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

Departing tangentially on a straight line: DEP LT

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line lies in the extension of the last contour element. P_N is separated from P_E by the distance LEN.

▶ Program the last contour element with the end point P_E and radius compensation.

▶ Initiate the dialog with the APPR/DEP key and DEP LT soft key:

[DEP LT] ▶ LEN: Enter the distance from the last contour element P_E to the end point P_N.
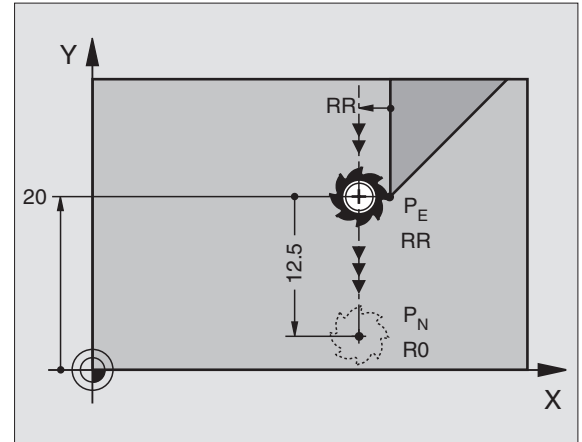
Resulting NC blocks

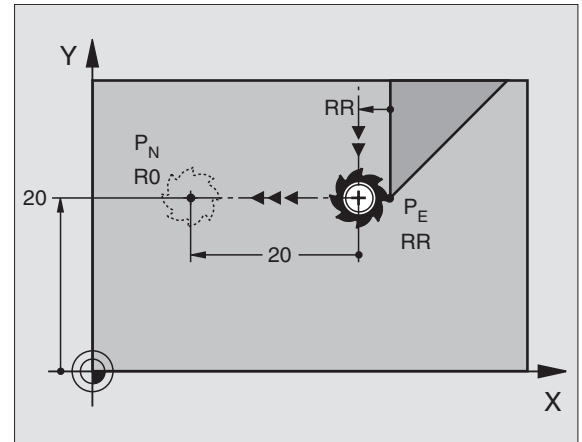|	
23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LT LEN 12.5 F100	Depart contour by LEN = 12.5 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

Departing on a straight line perpendicular to the last contour point: DEP LN

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line departs on a path that is perpendicular to the last contour point P_E. P_N is separated from P_E by the distance LEN + tool radius.

▶ Program the last contour element with the end point P_E and radius compensation.

▶ Initiate the dialog with the APPR/DEP key and DEP LN soft key:

[DEP LN] ▶ LEN: Enter the distance from the last contour element P_E to the end point P_N.
Important: Always enter LEN as a positive value!

Resulting NC blocks

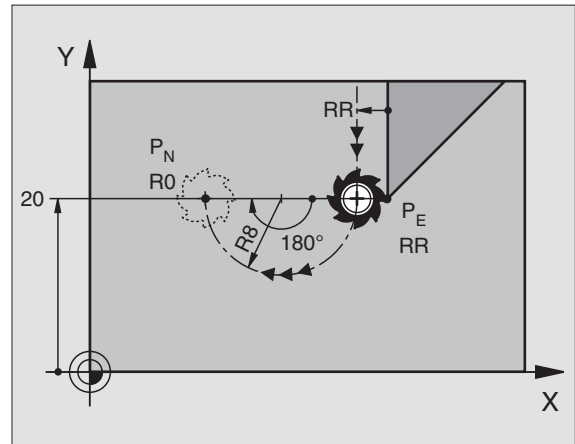|	
23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LN LEN+20 F100	Depart perpendicular to contour by LEN = 20 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

Departing tangentially on a circular arc: DEP CT

The tool moves on a circular arc from the last contour point P_E to the end point P_N. The arc is tangentially connected to the last contour element.

▶ Program the last contour element with the end point P_E and radius compensation.

▶ Initiate the dialog with the APPR/DEP key and DEP CT soft key:

| DEP CT | ▶ RADIUS R of the circular arc |

■ If the tool should depart the workpiece in the direction of the radius compensation (i.e. to the right with RR or to the left with RL):
Enter R as a positive value.

■ If the tool should depart the workpiece on the direction **opposite** to the radius compensation:
Enter R as a negative value.

▶ CENTER ANGLE CCA of the arc

Resulting NC blocks

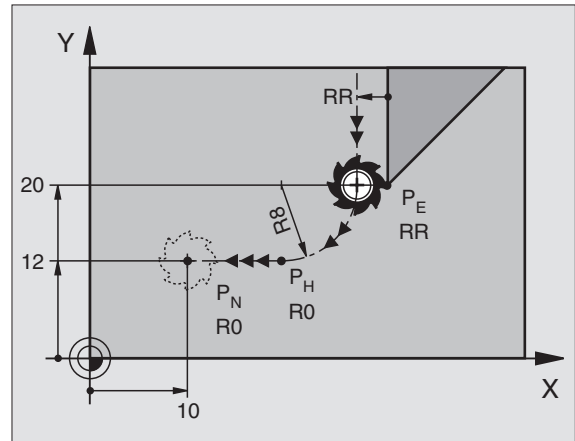| 23 L Y+20 RR F100 | Last contour element: PE with radius compensation |
| 24 DEP CT CCA 180 R+10 F100 | Center angle=180°, arc radius=10 mm |
| 25 L Z+100 FMAX M2 | Retract in Z, return to block 1, end program |

Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

The tool moves on a circular arc from the last contour point P_E to an auxiliary point P_H. It then moves from P_H to the end point P_N on a straight line. The arc is tangentially connected both to the last contour element and to the line from P_H to P_N. Once these lines are known, the radius R then suffices to completely define the tool path.

▶ Program the last contour element with the end point P_E and radius compensation.

▶ Initiate the dialog with the APPR/DEP key and DEP LCT soft key:

| DEP LCT | ▶ Enter the COORDINATES of the end point P_N. |

▶ RADIUS R of the arc
Always enter R as a positive value

Resulting NC blocks

23 L Y+20 RR F100	Last contour element: PE with radius compensation
24 DEP LCT X+10 Y+12 R+8 F100	Coordinates P_N, arc radius = 10 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

6.4 Path Contours — Cartesian Coordinates

Overview of path functions

Function	Path function key	Tool movement	Required input
Line		Straight line	Coordinates of the straight-line end point
CHam**F**er		Chamfer between two straight lines	Chamfer side length
Circle **C**enter		No tool movement	Coordinates of the circle center or pole
Circle		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circle by **R**adius		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circle **T**angential		Circular arc with tangential connection to the previous contour element	Coordinates of the arc end point
Corner **R**ou**ND**ing		Circular arc with tangential connection to the previous and subsequent contour elements	Rounding-off radius R

Straight line L

The tool moves on a straight line from its current position to the line end point. The starting point for the straight line is the end point that was programmed in the previous block.

▶ Enter the COORDINATES of the end point.

Further entries, if necessary:

▶ RADIUS COMPENSATION RL/RR/R0

▶ FEED RATE F

▶ MISCELLANEOUS FUNCTION M

Resulting NC blocks

```
7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10
```

Actual position capture
You can also generate a straight-line block (L block) by using the actual-position-capture key:

▶ In the MANUAL OPERATION mode, move the tool to the position you wish to capture.

▶ Switch the screen display to PROGRAMMING AND EDITING.

▶ Select the program block after which you want to insert the L block.

▶ Press the actual-position-capture key: The TNC generates an L block with the actual position coordinates.

Inserting a chamfer CHF between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

■ The blocks before and after the CHF block must be in the same working plane.

■ The radius compensation before and after the chamfer block must be the same.

■ An inside chamfer must be large enough to accommodate the current tool.

▶ CHAMFER SIDE LENGTH: Enter the length of the chamfer
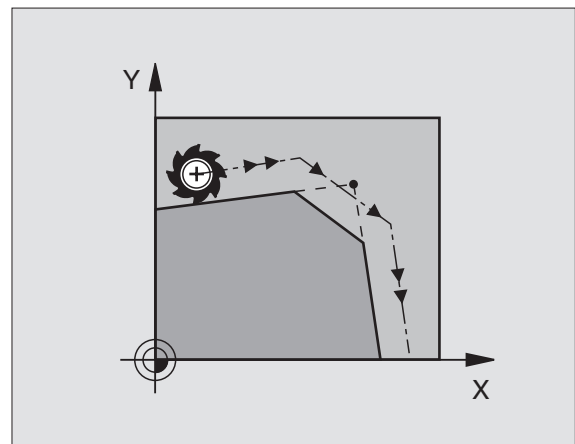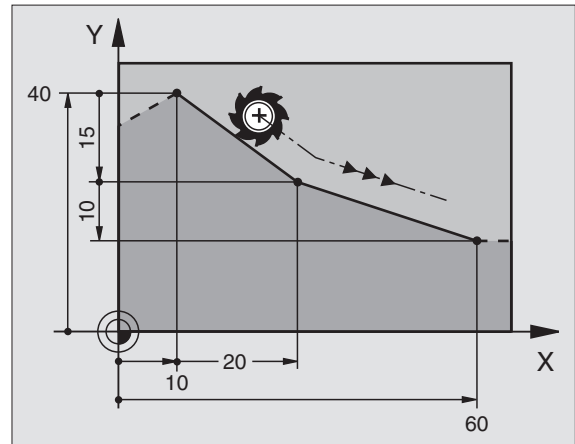
Please observe the notes on the next page!

Resulting NC blocks

```
7 L X+0 Y+30 RL F300 M3
```
```
8 L  X+40 IY+5
```
```
9 CHF 12
```
```
10 L  IX+5 Y+0
```

☞ You cannot start a contour with a CHF block!

A chamfer is possible only in the working plane.

The feed rate for chamfering is taken from the previous block.

The corner point is cut off by the chamfer and is not part of the contour.

Circle center CC

You can define a circle center CC for circles that are programmed with the C key (circular path C). This is done in the following ways:

■ Entering the Cartesian coordinates of the circle center

■ Using the circle center defined in an earlier block

■ Capturing the coordinates with the actual-position-capture key

▶ COORDINATES CC: Enter the circle center coordinates

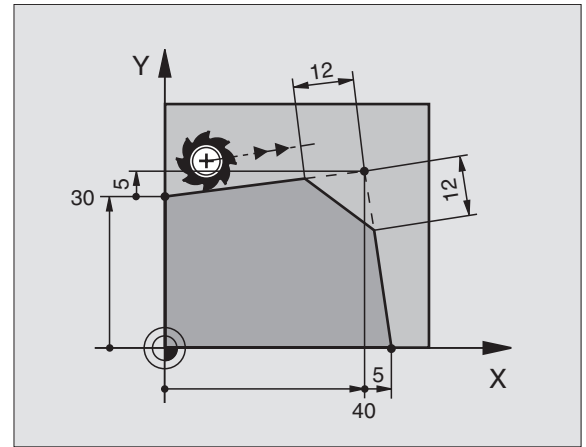If you want to capture the last programmed position: Do not enter any coordinates.

Resulting NC blocks

```
5 CC X+25 Y+25
```
or

```
10 L X+25 Y+25
```
```
11 CC
```

The program blocks 10 and 11 do not refer to the illustration.
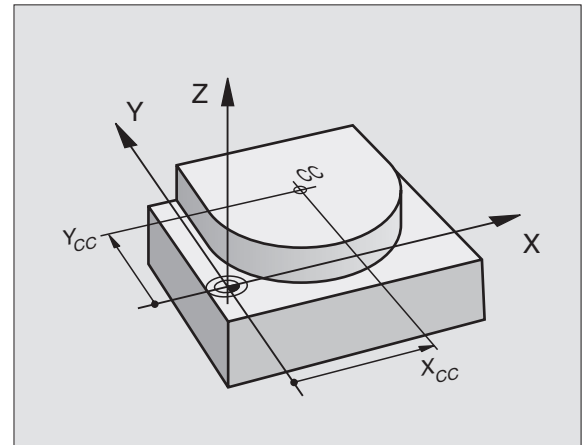
Duration of effect
The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Entering the circle center CC incrementally
If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

> The only effect of CC is to define a position as a circle center — the tool does not move to the position.
>
> The circle center also serves as the pole for polar coordinates.

Circular path C around circle center CC

Before programming a circular path C, you must first enter the circle center CC. The last programmed tool position before the C block is used as the circle starting point.

▶ Move the tool to the circle starting point.

▶ Enter the COORDINATES of the circle center.

▶ Enter the COORDINATES of the arc end point

▶ DIRECTION OF ROTATION DR

Further entries, if necessary:

▶ FEED RATE F

▶ MISCELLANEOUS FUNCTION M
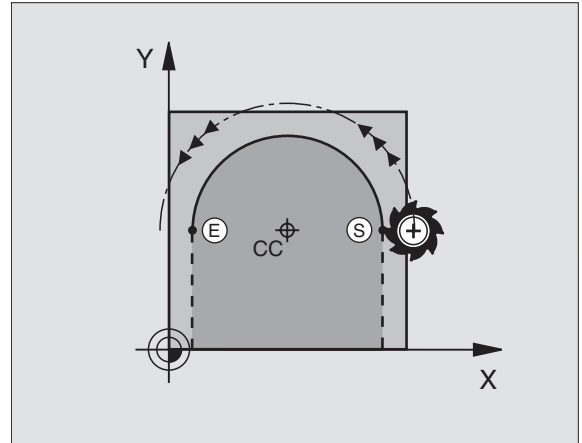
Resulting NC blocks

```
5 CC X+25 Y+25
```
```
6 L X+45 Y+25 RR F200 M3
```
```
7 C X+45 Y+25 DR+
```

Full circle
Enter the same point you used as the starting point for the end point in a C block.

> The starting and end points of the arc must lie on the circle.
>
> Input tolerance: up to 0.016 mm (selected with MP7431).

Circular path CR with defined radius

The tool moves on a circular path with the radius R.

▶ Enter the COORDINATES of the arc end point.

▶ RADIUS R
Note: The algebraic sign determines the size of the arc.

▶ DIRECTION OF ROTATION DR
Note: The algebraic sign determines whether the arc is concave or convex.
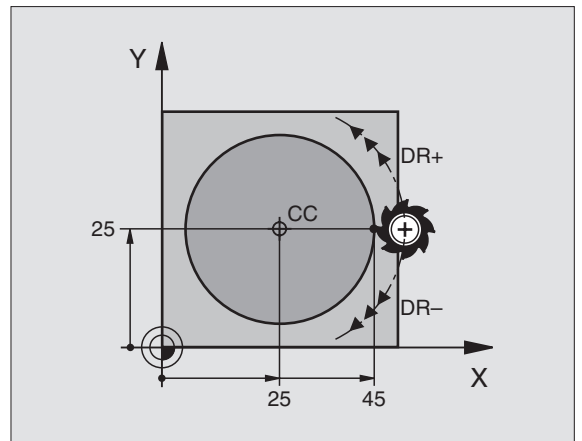
Further entries, if necessary:

▶ FEED RATE F

▶ MISCELLANEOUS FUNCTION M
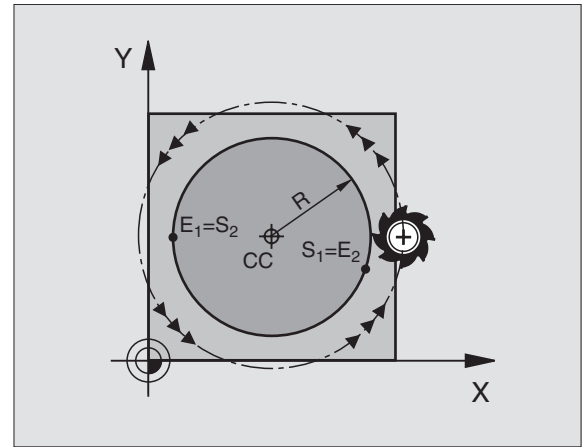
Full circle
For a full circle, program two CR blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first (see figure to the upper right).

Central angle CCA and arc radius R
The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

Larger arc: CCA>180°
Enter the radius with a negative sign R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation DR– (with radius compensation RL)

Concave: Direction of rotation DR+ (with radius compensation RL)

Resulting NC blocks
See figures at middle and lower right.

```
10  L  X+40  Y+40  RL  F200 M3
11  CR X+70  Y+40  R+20 DR– (arc 1)
```
or
```
11  CR X+70  Y+40  R+20 DR+ (arc 2)
```
or
```
11  CR X+70  Y+40  R-20 DR- (arc 3)
```
or
```
11  CR X+70  Y+40  R-20 DR+ (arc 4)
```

Please observe the notes on the next page!
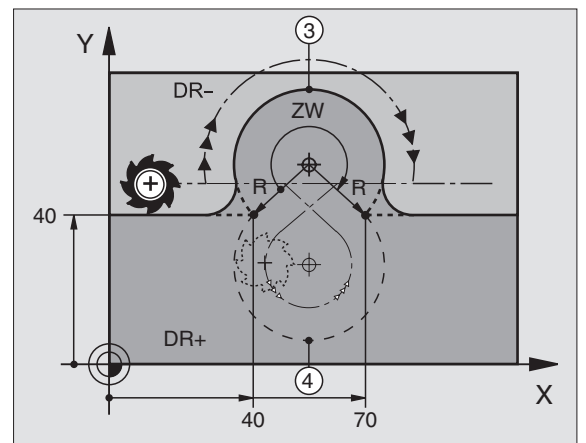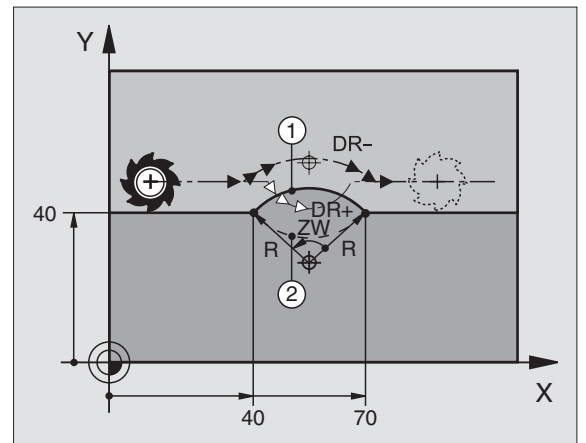
> The distance from the starting and end points of the arc
> diameter cannot be greater than the diameter of the arc.
>
> The maximum possible radius is 99.9999 m.
>
> You can also enter rotary axes A, B and C.

Circular path CT with tangential connection

The tool moves on an arc that starts at a tangent with the previously programmed contour element.

A transition between two contour elements is called "tangential" when there is no kink or corner at the intersection between the two contours — the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the CT block. This requires at least two positioning blocks.

▶ Enter the COORDINATES of the arc end point.

Further entries, if necessary:

▶ FEED RATE F

▶ MISCELLANEOUS FUNCTION M

Resulting NC blocks

```
7 L X+0 Y+25 RL F300 M3
8 L X+25 Y+30
9 CT X+45 Y+20
10 L Y+0
```

> A tangential arc is a two-dimensional operation: the
> coordinates in the CT block and in the contour element
> preceding it must be in the same plane of the arc.
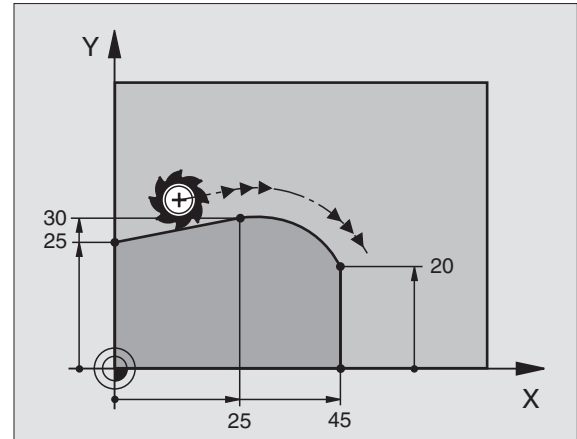
Corner Rounding RND

The RND function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be large enough to accommodate the tool.

▶ ROUNDING-OFF RADIUS: Enter the radius of the arc.

▶ FEED RATE for rounding the corner.

Resulting NC blocks

```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

☞ In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc.

The corner point is cut off by the rounding arc and is not part of the contour.

A feed rate programmed in the RND block is effective only in that block. After the RND block, the previous feed rate becomes effective again.

You can also use an RND block for a tangential contour approach if you do not want to use an APPR function.
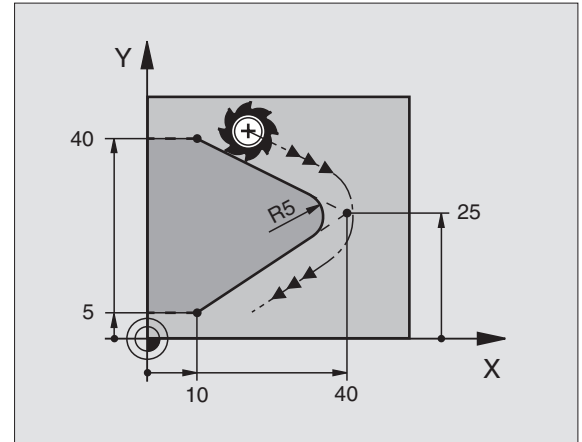
Example: Linear movements and chamfers with Cartesian coordinates
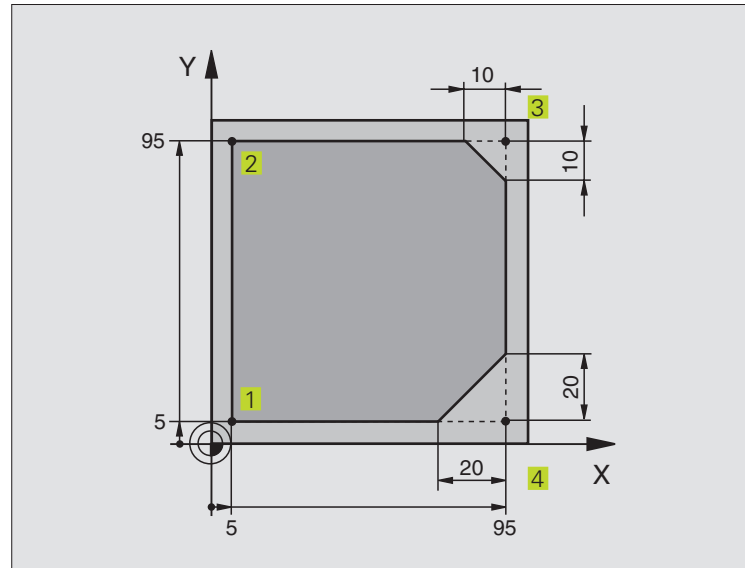
0	BEGIN PGM LINEAR MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define tool in the program
4	TOOL CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5	L Z+250 R0 F MAX	Retract tool in the spindle axis at rapid traverse FMAX
6	L X-10 Y-10 R0 F MAX	Pre-position the tool
7	L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8	APPR LT X+5 Y+5 LEN10 RL F300	Approach the contour at point 1 on a straight line with tangential connection
9	L Y+95	Move to point 2
10	L X+95	Point 3: first straight line for corner 3
11	CHF 10	Program chamfer with length 10 mm
12	L Y+5	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
13	CHF 20	Program chamfer with length 20 mm
14	L X+5	Move to last contour point 1, second straight line for corner 4
15	DEP LT LEN10 F1000	Depart the contour on a straight line with tangential connection
16	L Z+250 R0 F MAX M2	Retract the tool, end of program
17	END PGM LINEAR MM	

Example: Circular movements with Cartesian coordinates
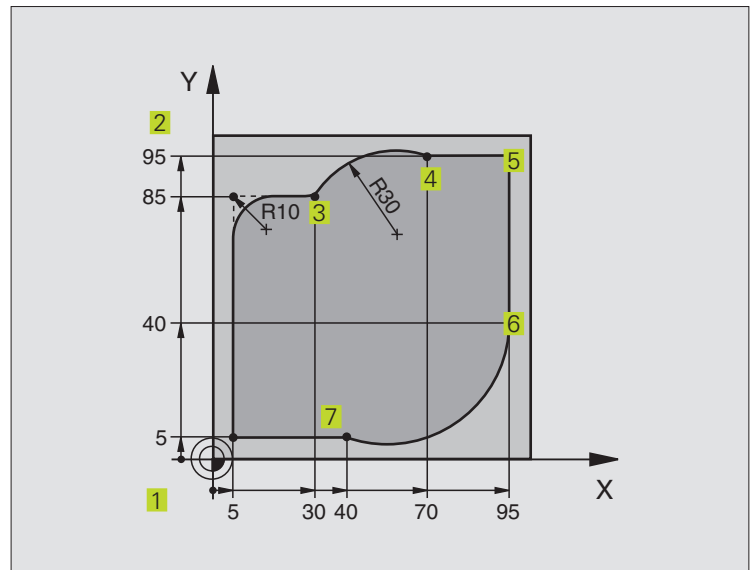
| 0	BEGIN PGM CIRCULAR MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define tool in the program
4	TOOL CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5	L Z+250 R0 F MAX	Retract tool in the spindle axis at rapid traverse FMAX
6	L X-10 Y-10 R0 F MAX	Pre-position the tool
7	L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8	APPR LCT X+5 Y+5 R5 RL F300	Approach the contour at point 1 on a circular arc with tangential connection
9	L Y+85	Point 2: first straight line for corner 2
10	RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
11	L X+30	Move to point 3: Starting point of the arc with CR
12	CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
13	L X+95	Move to point 5
14	L Y+40	Move to point 6
15	CT X+40 Y+5	Move to point 7: End point of the arc, circular arc with tangential connection to point 6, TNC automatically calculates radius
16	L X+5	Move to last contour point 1
17	DEP LCT X-20 Y-20 R5 F1000	Depart the contour on a circular arc with tangential connection
18	L Z+250 R0 F MAX M2	Retract the tool, end of program
19	END PGM CIRCULAR MM	

Example: Full circle with Cartesian coordinates
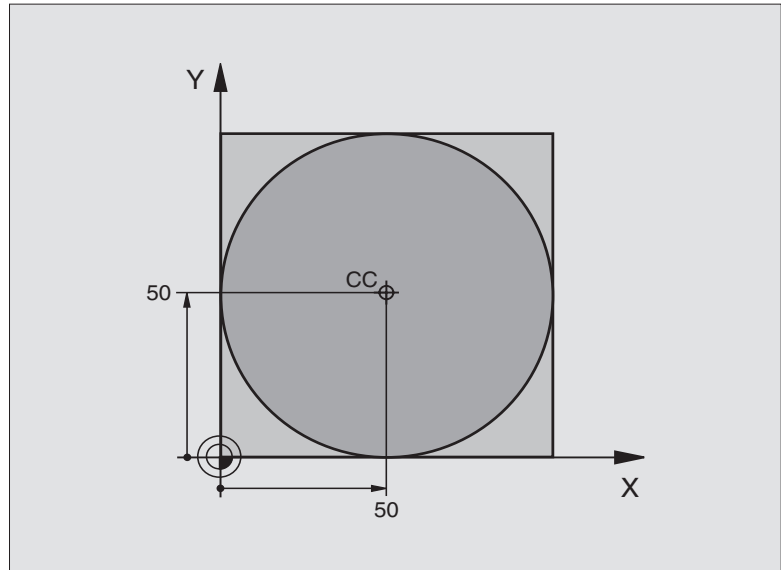
0	BEGIN PGM C-CC MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+12.5	Define the tool
4	TOOL CALL 1 Z S3150	Call the tool
5	CC X+50 Y+50	Define the circle center
6	L Z+250 R0 F MAX	Retract the tool
7	L X-40 Y+50 R0 F MAX	Pre-position the tool
8	L Z-5 R0 F1000 M3	Move to working depth
9	APPR LCT X+0 Y+50 R5 RL F300	Approach the starting point of the circle on a circular arc with tangential connection
10	C X+0 DR-	Move to the circle end point (= circle starting point)
11	DEP LCT X-40 Y+50 R5 F1000	Depart the contour on a circular arc with tangential connection
12	L Z+250 R0 F MAX M2	Retract the tool, end of program
13	END PGM CCC MM	

6.5 Path Contours — Polar Coordinates

With polar coordinate you can define a position in terms of its angle PA and its distance PR relative to a previously defined pole CC (see section 4.1 "Fundamentals of NC").

Polar coordinates are useful with:

■ Positions on circular arcs

■ Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

Function	Path function keys	Tool movement	Required input
Line **LP**	[⌐✎] + [P]	Straight line	Polar radius, polar angle of the straight-line end point
Circular arc **CP**	[⌐ᶜ] + [P]	Circular path around circle center/ pole CC to arc end point	Polar angle of the arc end point, direction of rotation
Circular arc **CTP**	[CTᴘ] + [P]	Circular path with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helical interpolation	[⌐ᶜ] + [P]	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis

Polar coordinate origin: Pole CC

You can define the pole CC anywhere in the part program before blocks containing polar coordinates. Enter the pole in Cartesian coordinates as a circle center in a CC block.

[cc ⊕] ▶ COORDINATES CC: Enter Cartesian coordinates for the pole, or:

If you want to use the last programmed position: do not enter any coordinates
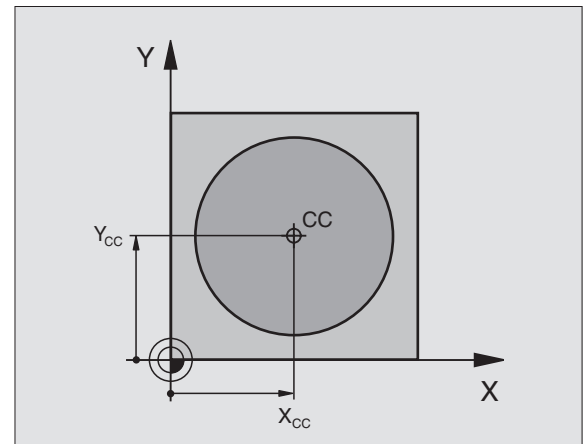
Straight line LP

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point that was programmed in the preceding block.

⌐P ▸ POLAR COORDINATES-RADIUS PR: Enter the distance from the pole CC to the straight-line end point.

▸ POLAR COORDINATES-ANGLE PA: Angular position of the straight-line end point between –360° and +360°

The sign of PA depends on the angle reference axis:
Angle from angle reference axis to PR is counterclockwise: PA>0
Angle from angle reference axis to PR is clockwise: PA<0
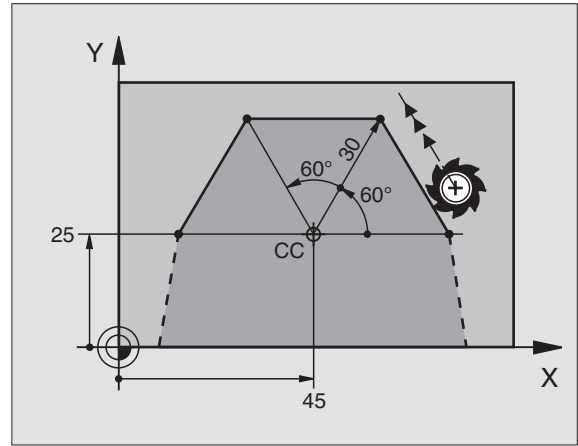
Resulting NC blocks

```
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180
```

Circular path CP around pole CC

The polar coordinate radius PR is also the radius of the arc. It is defined by the distance from the starting point to the pole CC. The last programmed tool position before the CP block is the starting point of the arc.

⌐P ▸ POLAR COORDINATES-ANGLE PA: Angular position of the arc end point between –5400° and +5400°

▸ DIRECTION OF ROTATION DR

Resulting NC blocks

```
18 CC X+25 Y+25
19 LP PR+20 PA+0 RR F250 M3
20 CP PA+180 DR+
```
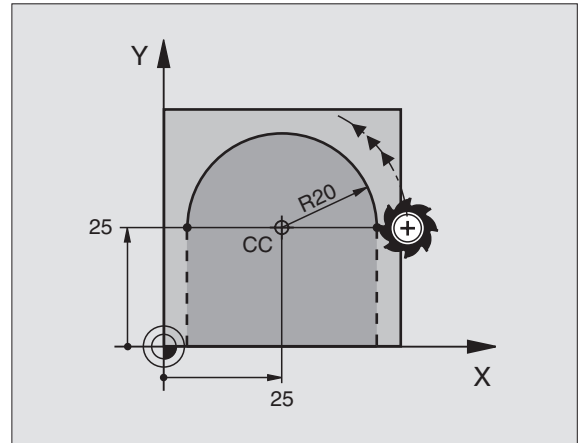
☞ For incremental coordinates, enter the same sign for DR and PA.

Circular path CTP with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.

[CTP] [P] ▶ POLAR COORDINATES-RADIUS PR: Distance from the arc end point to the pole CC.

▶ POLAR COORDINATES-ANGLE PA: Angular position of the arc end point.

Resulting NC blocks

```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

☞ The pole CC is **not** the center of the contour arc!
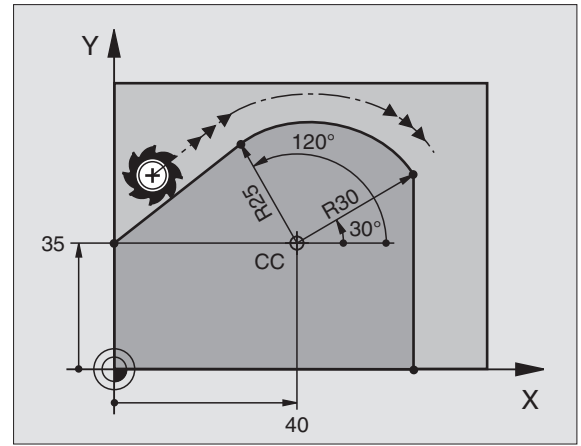
Helical interpolation

A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane.

A helix is programmed only in polar coordinates.

Applications

■ Large-diameter internal and external threads

■ Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in a upward direction, you need the following data:

Thread revolutions n	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P x thread revolutions n
Incremental total angle IPA	Thread revolutions x 360°
Starting angle IPA	Angle for start of thread + angle for thread overrun
Starting coordinate Z	Thread pitch P x (thread revolutions + thread overrun at start of thread)
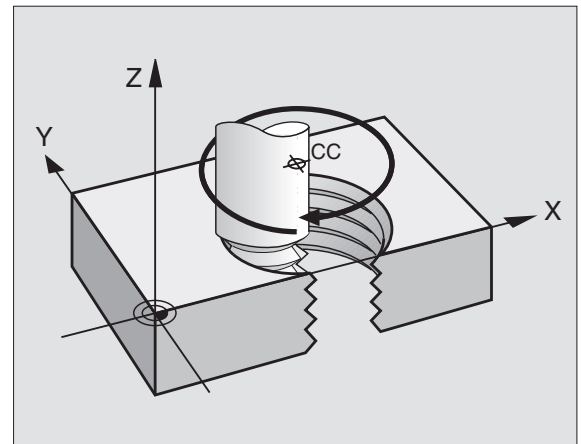
Shape of the helix

The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed	Z+	DR+	RL
Left-handed	Z+	DR−	RR
Right-handed	Z−	DR−	RR
Left-handed	Z−	DR+	RL
External thread			
Right-handed	Z+	DR+	RR
Left-handed	Z+	DR−	RL
Right-handed	Z−	DR−	RL
Left-handed	Z−	DR+	RR

Programming a helix

☞ Always enter the same algebraic sign for the direction of rotation DR and the incremental total angle IPA. The tool may otherwise move in a wrong path and damage the contour.

For the total angle IPA, you can enter a value from −5400° to +5400°. If the thread has of more than 15 revolutions, program the helix in a program section repeat (see section 9.2 "Program Section Repeats").

▶ POLAR COORDINATES-ANGLE: Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, identify the tool axis with an axis selection key.**

▶ Enter the COORDINATE for the height of the helix in incremental dimensions.

▶ Direction of rotation DR
Clockwise helix: DR−
Counterclockwise helix: DR+

▶ RADIUS COMPENSATION RL/RR/0
Enter the radius compensation according to the table above.
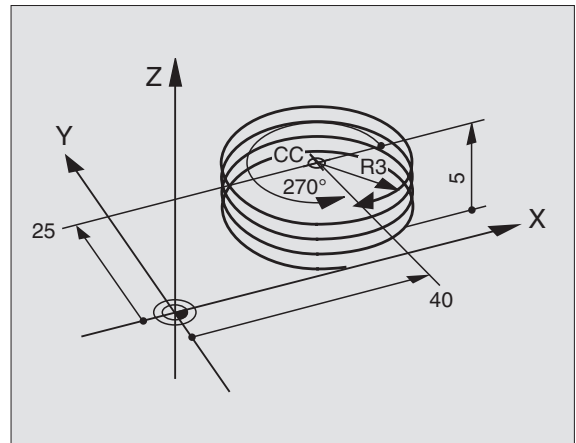
Resulting NC blocks

```
12 CC X+40 Y+25
```
```
13 Z+0 F100 M3
```
```
14 LP PR+3 PA+270
```
```
15 CP IPA−1800 IZ+5 DR− RL F50
```

Example: Linear movement with polar coordinates
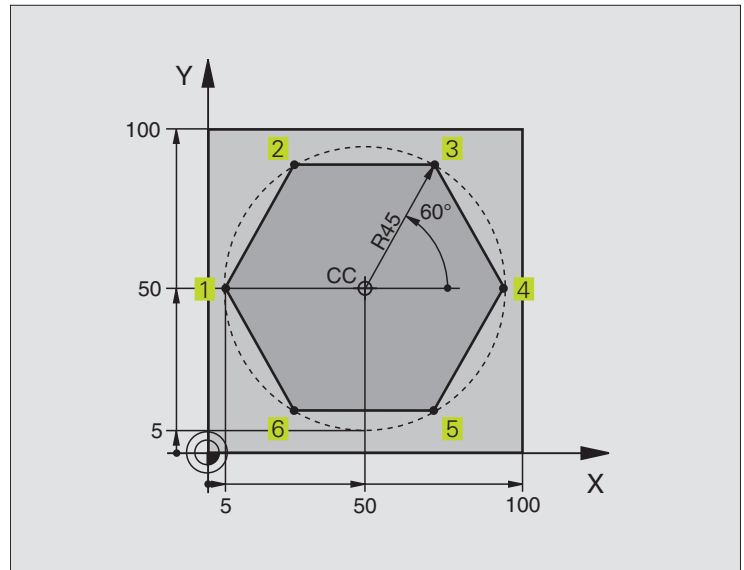
0	BEGIN PGM LINEARPO MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+7.5	Define the tool
4	TOOL CALL 1 Z S4000	Call the tool
5	CC X+50 Y+50	Define the datum for polar coordinates
6	L Z+250 R0 F MAX	Retract the tool
7	LP PR+60 PA+180 R0 F MAX	Pre-position the tool
8	L Z-5 R0 F1000 M3	Move to working depth
9	APPR PLCT PR+45 PA+180 R5 RL F250	Approach the contour at point 1 on a circular arc with tangential connection
		Move to point 2
10	LP PA+120	Move to point 3
11	LP PA+60	Move to point 4
12	LP PA+0	Move to point 5
13	LP PA-60	Move to point 6
14	LP PA-120	Move to point 1
15	LP PA+180	Depart the contour on a circular arc with tangential connection
16	DEP PLCT PR+60 PA+180 R5 F1000	Retract the tool, end of program
17	L Z+250 R0 F MAX M2	
18	END PGM LINEARPO MM	

Example: Helix
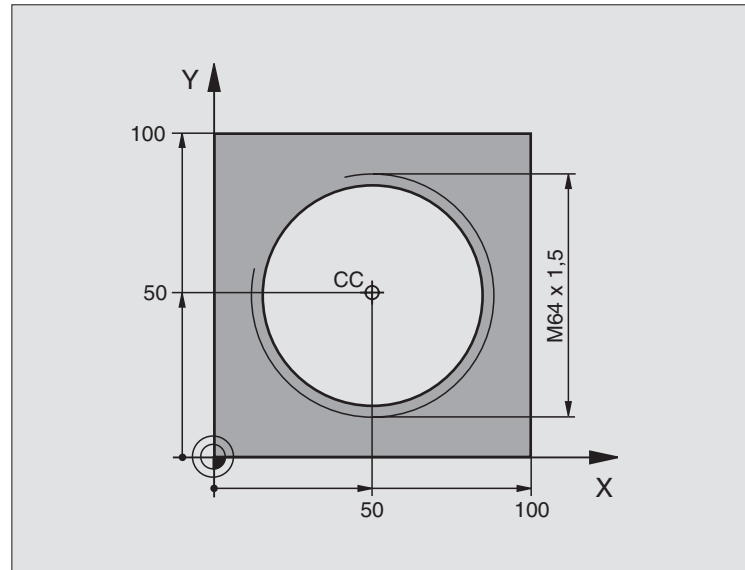
`0   BEGIN PGM HELIX MM`	
`1   BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2   BLK FORM 0.2 X+100 Y+100 Z+0`	
`3   TOOL DEF 1 L+0 R+5`	Define the tool
`4   TOOL CALL 1 Z S1400`	Call the tool
`5   L Z+250 R0 F MAX`	Retract the tool
`6   L X+50 Y+50 R0 F MAX`	Pre-position the tool
`7   CC`	Transfer the last programmed position as the pole
`8   L Z-12.75 R0 F1000 M3`	Move to working depth
`9   APPR PCT PR+32 PA-180 CCA180` `R+2`	Approach the contour on a circular arc with tangential connection
`    RL F100`	Helical interpolation
`10  CP IPA+3240 IZ+13.5 DR+ F200`	Depart the contour on a circular arc with tangential connection
`11  DEP CT CCA180 R+2`	Retract the tool, end of program
`12  L Z+250 R0 F MAX M2`	
`13  END PGM HELIX MM`	

To cut a thread with more than 16 revolutions:

`...`	
`8   L Z-12.75 R0 F1000`	
`9   APPR PCT PR+32 PA-180 CCA180 R+2 RL F100`	Identify beginning of program section repeat
`10  LBL 1`	Enter the thread pitch as an incremental IZ dimension
`11  CP IPA+360 IZ+1.5 DR+ F200`	Program the number of repeats (thread revolutions)
`12  CALL LBL 1 REP 24`	
`13  DEP CT CCA180 R+2`	

6.6 Path Contours — FK Free Contour Programming

Fundamentals

Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys. You may, for example, only have the following data on a specific contour element:

■ Known coordinates on the contour element or in its proximity

■ Coordinate data that are referenced to another contour element

■ Directional data and data regarding the course of the contour

You can enter such dimensional data directly by using the FK free contour programming function. The TNC derives the contour from the known coordinate data and supports the programming dialog with the interactive programming graphics. The figure to the upper right shows a workpiece drawing for which FK programming is the most convenient programming method.
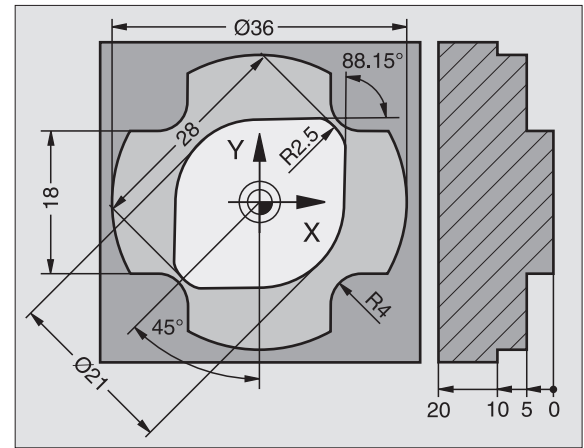
Graphics during FK programming
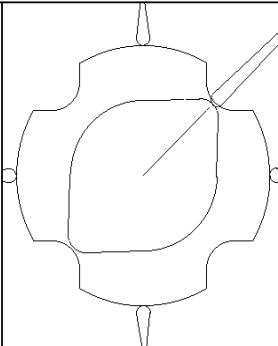
Incomplete coordinate data often are not sufficient to fully define a workpiece contour. In this case, the TNC indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing. The FK graphic displays the elements of the workpiece contour in different colors:

White The contour element is fully defined.

Green The entered data describe a limited number of possible solutions: select the correct one.

Red The entered data are not sufficient to determine the contour element: enter further data.

If the entered data permit a limited number of possible solutions and the contour element is displayed in green, select the correct contour element as follows:

SHOW ▶ Press the SHOW soft key repeatedly until the correct contour element is displayed.

FSELECT ▶ If the displayed contour element matches the drawing, select the contour element with FSELECT.

Select the green contour elements as soon as possible with the FSELECT soft key. In this way, you can reduce the ambiguity of subsequent elements.

If you do not yet wish to select a green contour element, press the EDIT soft key to continue the FK dialog.

☞ The machine tool builder may use other colors for the FK graphics.

NC blocks from a program that you called with PGM CALL are displayed in another color.

Initiating the FK dialog

While you are writing a part program, the TNC displays the soft keys with which you can initiate the FK dialog (see table at right).

If you initiate the FK dialog with one of these soft keys, the TNC shows additional soft-key rows that you can use for entering known coordinates, directional data and data regarding the course of the contour.

☞ **The following prerequisites for FK programming must be observed:**

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane. The working plane is defined in the first BLK FORM block of the part program.

You must enter all available data for every contour element. Even the data that does not change must be entered in every block — otherwise it will not be recognized.

You cannot use Q parameters.

If both FK blocks and conventional blocks are entered in a program, the FK contour must be fully defined before you can return to conventional programming.

The TNC needs a fixed point from which it can calculate the contour elements. Use the gray path function keys to program a position that contains both coordinates of the working plane immediately before programming the FK contour. Do not enter any Q parameters in this block.

If the first block of an FK contour is an FCT or FLT block, you must program at least two NC block with the gray path function keys to fully define the direction of contour approach.

Do not program an FK contour immediately after an LBL label.

Contour element	Soft key
Straight line with tangential connection	FLT
Straight line without tangential connection	FL
Circular arc with tangential connection	FCT
Circular arc without tangential connection	FC

Free programming of straight lines

▶ To initiate the dialog for free programming of straight lines, press the FL soft key. The TNC displays additional soft keys — see table at right.

▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data are entered. If the entered data describe several solutions, the graphic will display the contour element in green (see "Graphics during FK programming").

See next page for the resulting NC blocks.

Straight line with tangential connection
If the straight line connects tangentially to another contour element, initiate the dialog with the FLT soft key:

▶ To initiate the dialog, press the FLT soft key.

▶ Use the soft keys (listed in the table at right) to enter all known data in the block.

Known data	Soft key
X coordinate of the straight-line end point	
Y coordinate of the straight-line end point	
Polar coordinate radius	
Polar coordinate angle	
Length of a straight line	
Gradient angle of a straight line	
Beginning or end of a closed contour	

For data that are based on other blocks see "Relative data," for auxiliary points see "Auxiliary points" in this section.

Free programming of circular arcs

▶ To initiate the dialog for free programming of circular arcs, press the FC soft key. The TNC displays soft keys with which you can enter direct data on the circular arc or data on the circle center — see table at right.

▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data are entered. If the entered data describe several solutions, the graphic will display the contour element in green (see "Graphics during FK programming").

Circular arc with tangential connection
If the circular arc connects tangentially to another contour element, initiate the dialog with the FCT soft key:

▶ To initiate the dialog, press the FCT soft key.

▶ Use the soft keys (listed in the table at right) to enter all known data in the block.

Direct data on a circular arc	Soft key
X coordinate of the arc end point	
Y coordinate of the arc end point	
Polar coordinate radius	
Polar coordinate angle	
Rotational direction of the arc	
Radius of the arc	
Angle of the leading axis to the arc end point	

6 Programming: Programming Contours

Gradient angle of an arc

The gradient angle AN of an arc is the angle of the entry tangent (see figure at right).

Chord length of an arc

The chord length LEN of an arc is the linear distance LEN between its end points (see figure at right).

Centers of free-programmed circles

The TNC calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an FK program block.

A circle center that was calculated or programmed conventionally is then no longer valid as a pole or circle center for the new FK contour: If you enter conventional polar coordinates that refer to a pole from a CC block you have defined previously, then you must enter the pole again in an FPOL block.

FPOL is entered in Cartesian coordinates and remains in effect until the TNC encounters a block in which another FPOL is defined.

Resulting NC blocks for FL, FPOL and FCT

```
7 FPOL X+20 Y+30
```
```
8 FL IX+10 Y+20 RR F100
```
```
9 FCT PR+15 IPA+30 DR+ R15
```

See figure at lower right.

Data on the circle center	Soft key
X coordinate of the circle center	CCX
Y coordinate of the circle center	CCY
Polar coordinate radius of the circle center	CC PR
Polar coordinate angle of the circle center	CC PA

For data that are based on other blocks see "Relative data," for auxiliary points see "Auxiliary points" in this section.
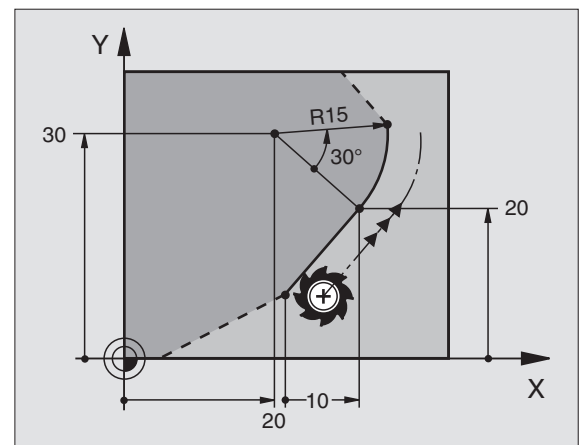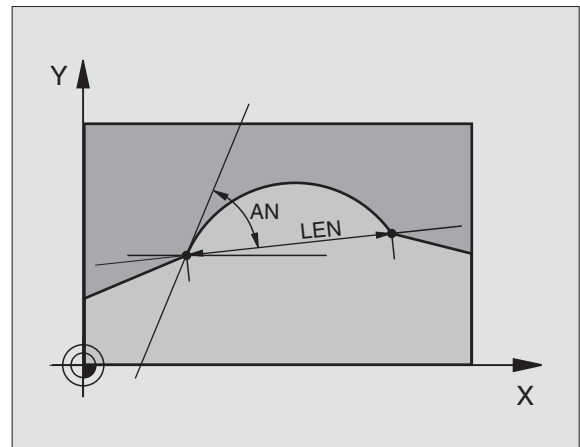
Auxiliary points

You can enter the coordinates of auxiliary points that are located on the contour or in its proximity for both free-programmed straight lines and free-programmed circular arcs. The soft keys are available as soon as you initiate the FK dialog with the FL, FLT, FC or FCT soft key.

Auxiliary points for a straight line
If the auxiliary points are located on a straight line or on the extension of a straight line, use the soft keys listed in the table at upper right.

If the auxiliary points are located at a distance D from a straight line, use the soft keys listed in the second table at right.

Auxiliary points for a circular arc
For circular arcs, you can enter 1,2 or 3 auxiliary points on the contour. The available soft keys are listed in the table at lower right.

Resulting NC blocks

```
13 FC DR– R10 P1X+42.929 P1Y+60.071
```
```
14 FLT AN-70 PDX+50 PDY+53 D10
```

See figure at lower right.

Auxiliary points on a straight line	Soft key
X coordinate auxiliary point P1 or P2	
Y coordinate auxiliary point P1 or P2	

Auxiliary points near a straight line	Soft key
X coordinate of an auxiliary point	
Y coordinate of an auxiliary point	
Distance auxiliary point/straight line	

Auxiliary points on a circular arc	Soft key
X coordinate of an auxiliary point P1, P2 or P3	
Y coordinate of an auxiliary point P1, P2 or P3	
Coordinates of an auxiliary point in the proximity of a circular arc	
Distance from an auxiliary point to a circular arc	
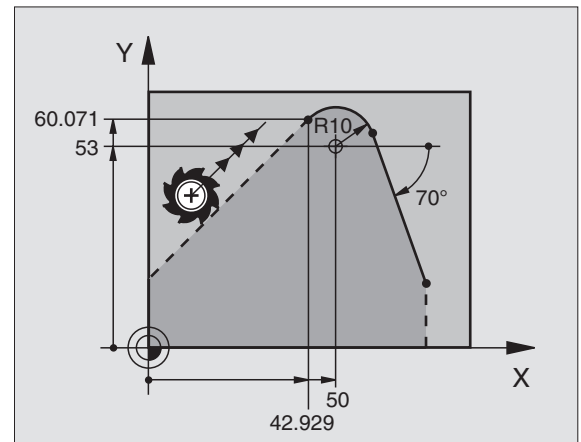
Relative data

Data whose values are based on another contour element are called relative data. The soft keys and program words for such entries begin with the letter **R** for **R**elative. The figure at right shows the entries that should be programmed as relative data.

The coordinates and angles for relative data are always programmed in **incremental** dimensions. You must also enter the block number of the contour element on which the data are based.

☞ The block number of the contour element on which the relative data are based can only be located up to 64 positioning blocks before the block in which you program the reference.

If you delete a block on which relative data are based, the TNC will display an error message. Change the program first before you clear this message.
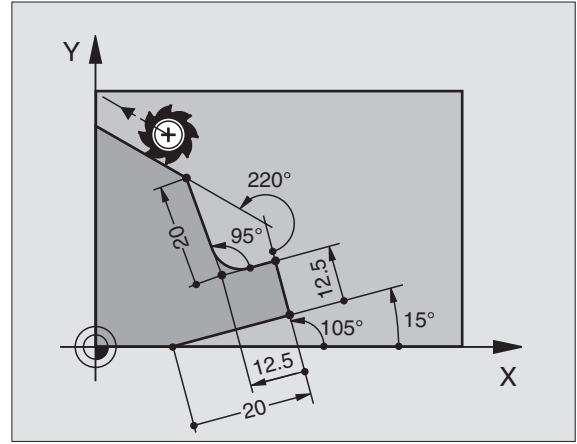
Relative data for a free-programmed straight line	Soft key
Coordinate relative to an end point of block N	RXⓃ RVⓃ
Change in the polar coordinate radius relative to block N	RPRⓃ
Change in the polar coordinate angle relative to block N	RPAⓃ
Angle between a straight line and another element	RANⓃ
Straight line parallel to another contour element	PARⓃ
Distance from a straight line to a parallel contour element	DP

Relative data for coordinates of an arc	Soft key
Coordinates relative to an end point of block N	RXⓃ RVⓃ
Change in the polar coordinate radius relative to block N	RPRⓃ
Change in the polar coordinate angle relative to block N	RPAⓃ
Angle between the entry tangent of the arc and another contour element	RANⓃ

Relative data for circle center coordinates
Soft key

CC coordinates relative to an end point of block N

`RCCX N` `RCCY N`

Change in the polar coordinate radius relative to block N

`RCCPR N`

Change in the polar coordinate angle relative to block N

`RCCPA N`

Resulting NC blocks
The known coordinates are relative to block N (see figure at upper right):

```
12 FPOL X+10 Y+10
13 FL PR+20 PA+20
14 FL AN+45
15 FCT IX+20 DR— R20 CCA+90 RX 13
16 FL IPR+35 PA+0 RPR 13
```

The known direction and the known distance from the contour element are relative to block N (see figure at center right):

```
17 FL LEN 20 AN+15
18 FL AN+105 LEN 12.5
19 FL PAR 17 DP 12.5
20 FSELECT 2
21 FL LEN 20 IAN+95
22 FL IAN+220 RAN 18
```


The known circle center coordinates are relative to block N (see figure at lower right):

```
12 FL X+10 Y+10 RL
13 FL ...
14 FL X+18 Y+35
15 FL ...
16 FL ...
17 FC DR— R10 CCA+0 ICCX+20 ICCY-15 RCCX12 RCCY14
```
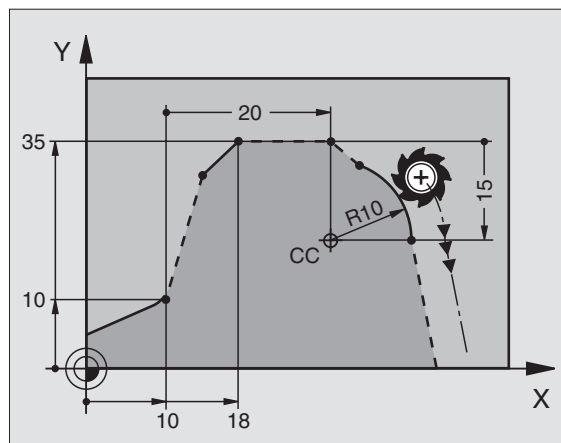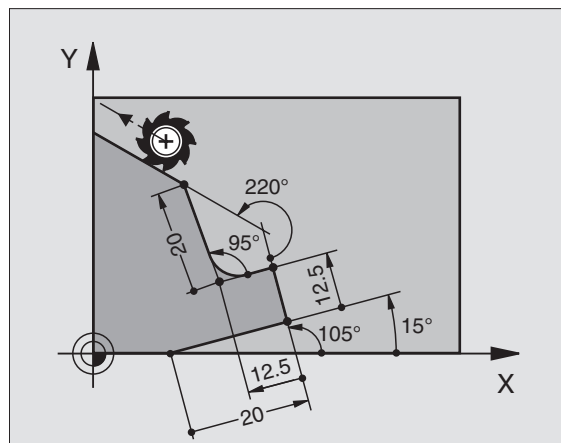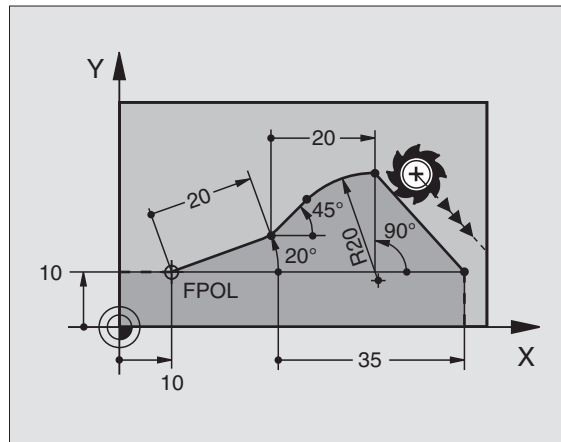
Closed contours

You can identify the beginning and end of a closed contour with the CLSD soft key. This reduces the number of possible solutions for the last contour element.

Enter CLSD as an addition to another contour data entry.

Converting FK programs

You can convert an FK program into HEIDENHAIN conversational format by using the file manager:

▶ Call the file manager and display the files.

▶ Move the highlight to the file you wish to convert.

> CONVERT
> FK->H

▶ Press the soft key MORE FUNCTIONS and then CONVERT FK->H. The TNC converts all FK blocks into HEIDENHAIN dialog blocks.

☞ Circle centers that you have entered before programming an FK contour may need to be redefined in the converted program. We recommend that you test the converted part program before executing it.
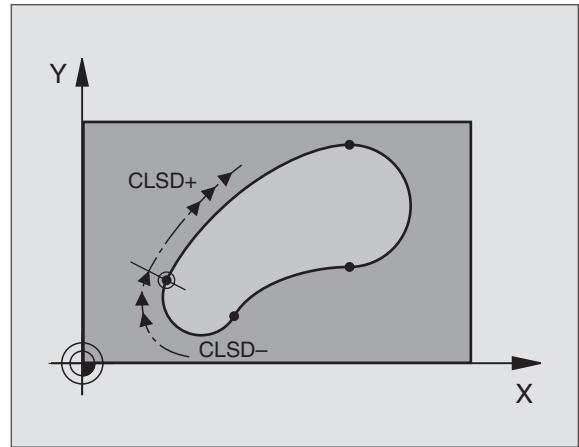
Example: FK programming 1
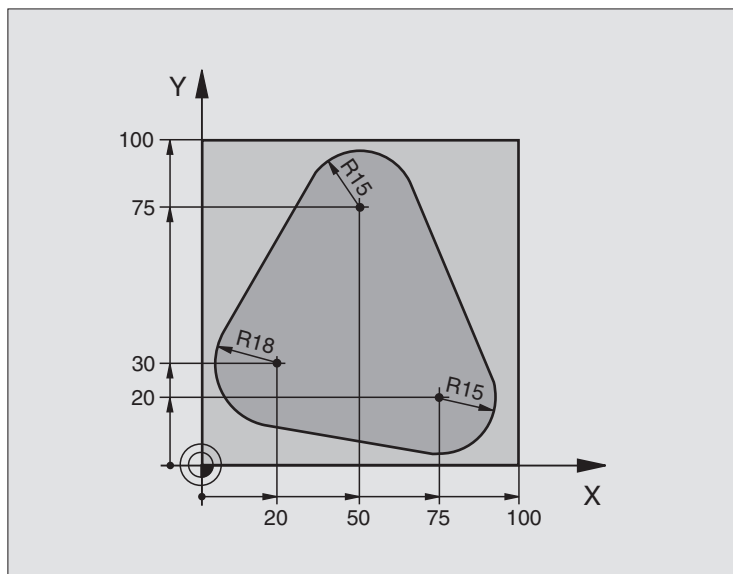
0	BEGIN PGM FK1 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define the tool
4	TOOL CALL 1 Z S500	Call the tool
5	L Z+250 R0 F MAX	Retract the tool
6	L X-20 Y+30 R0 F MAX	Pre-position the tool
7	L Z-10 R0 F1000 M3	Move to working depth
8	APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9	FC DR- R18 CLSD+ CCX+20 CCY+30	FK contour:
10	FLT	Program all known data for each contour element
11	FCT DR- R15 CCX+50 CCY+75	
12	FLT	
13	FCT DR- R15 CCX+75 CCY+20	
14	FLT	
15	FCT DR- R18 CLSD- CCX+20 CCY+30	
16	DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
17	L X-30 Y+0 R0 F MAX	
18	L Z+250 R0 F MAX M2	Retract the tool, end of program
19	END PGM FK1 MM	

Example: FK programming 2
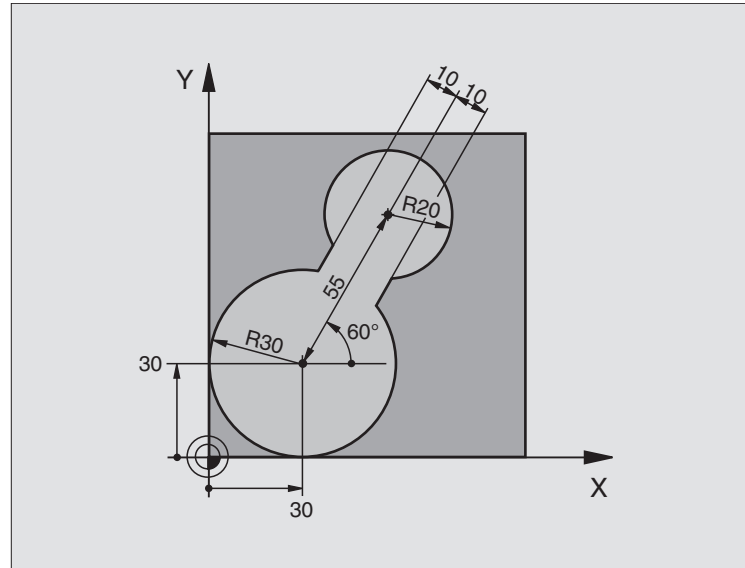
0	BEGIN PGM FK2 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+2	Define the tool
4	TOOL CALL 1 Z S4000	Call the tool
5	L Z+250 R0 F MAX	Retract the tool
6	L X+30 Y+30 R0 F MAX	Pre-position the tool
7	L Z+5 R0 F MAX M3	Pre-position the tool in the tool axis
8	L Z-5 R0 F100	Move to working depth
9	APPR LCT X+0 Y+30 R5 RL F350	Approach the contour on a circular arc with tangential connection
10	FPOL X+30 Y+30	FK contour:
11	FC DR- R30 CCX+30 CCY+30	Program all known data for each contour element
12	FL AN+60 PDX+30 PDY+30 D10	
13	FSELECT 3	
14	FC DR- R20 CCPR+55 CCPA+60	
15	FSELECT 2	
16	FL AN-120 PDX+30 PDY+30 D10	
17	FSELECT 3	
18	FC X+0 DR- R30 CCX+30 CCY+30	
19	FSELECT 2	
20	DEP LCT X+30 Y+30 R5	Depart the contour on a circular arc with tangential connection
21	L Z+250 R0 F MAX M2	Retract the tool, end of program
22	END PGM FK2 MM	

Example: FK programming 3
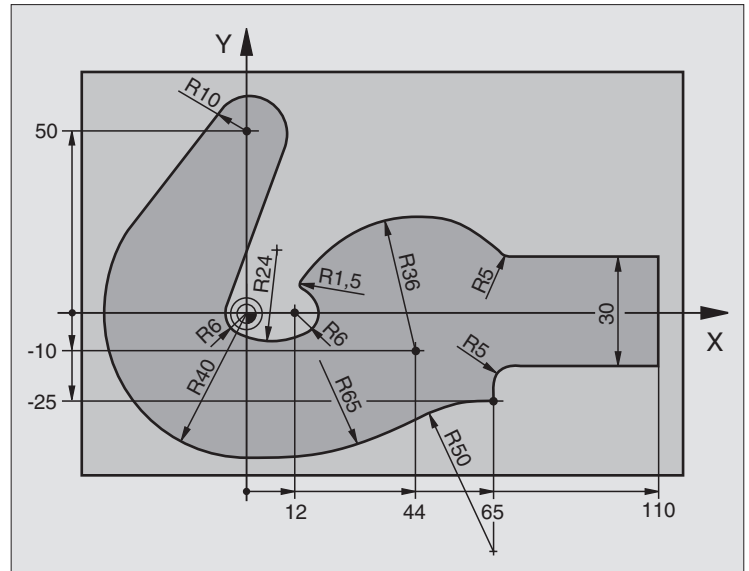
0	BEGIN PGM FK3 MM	
1	BLK FORM 0.1 Z X-45 Y-45 Z-20	Define the blank form
2	BLK FORM 0.2 X+120 Y+70 Z+0	
3	TOOL DEF 1 L+0 R+3	Define the tool
4	TOOL CALL 1 Z S4500	Call the tool
5	L Z+250 R0 F MAX	Retract the tool
6	L X-70 Y+0 R0 F MAX	Pre-position the tool
7	L Z-5 R0 F1000 M3	Move to working depth
8	APPR CT X-40 Y+0 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9	FC DR- R40 CCX+0 CCY+0	FK contour:
10	FLT	Program all known data for each contour element
11	FCT DR- R10 CCX+0 CCY+50	
12	FLT	
13	FCT DR+ R6 CCX+0 CCY+0	
14	FCT DR+ R24	
15	FCT DR+ R6 CCX+12 CCY+0	
16	FSELECT 2	
17	FCT DR- R1.5	
18	FCT DR- R36 CCX+44 CCY-10	
19	FSELECT 2	
20	FCT DR+ R5	
21	FLT X+110 Y+15 AN+0	
22	FL AN-90	

```
23    FL X+65 AN+180 PAR21 DP30
24    RND R5
25    FL X+65 Y-25 AN-90
26    FC DR+ R50 CCX+65 CCY-75
27    FCT DR- R65
28    FSELECT 1
29    FCT Y+0 DR- R40 CCX+0 CCY+0
30    FSELECT 4
31    DEP CT CCA90 R+5 F1000          Depart the contour on a circular arc with tangential connection
32    L X-70 R0 F MAX
33    L Z+250 R0 F MAX M2             Retract the tool, end of program
34    END PGM FK3 MM
```

7

Programming:

Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and STOP

With the TNC's miscellaneous functions — also called M functions — you can affect:

- Program run, such as a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring behavior of the tool

> The machine tool builder may add some M functions that are not described in this User's Manual. Refer to your machine manual for more information.

M functions are always entered at the end of a positioning block. The TNC then displays the following dialog question:

`MISCELLANEOUS FUNCTION M ?`

You usually only enter the number of the M function in the programming dialog. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the MANUAL OPERATION and ELECTRONIC HANDWHEEL modes of operation, the M functions are entered with the M soft key.

Please note that some F functions become effective at the start of a positioning block, and other at the end.

M functions come into effect in the block in which they are called. Unless the M function is only effective blockwise, it is canceled in a subsequent block or at the end of program. Some M functions are effective only in the block in which they are called.

Entering an M function in a STOP block
If you program a STOP block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a STOP block:

STOP ▶ To program an interruption of program run, press the STOP key.

▶ Enter a MISCELLANEOUS FUNCTION M.

Resulting NC block
`87 STOP M6`

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

M	Effect	Effective at
M00	Stop program run Spindle STOP Coolant OFF	Block end
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear status display (depending on machine parameter)	Block end
M03	Spindle ON clockwise	Block start
M04	Spindle ON counterclockwise	Block start
M05	Spindle STOP	Block end
M06	Tool change Spindle STOP Stop program run (depending on machine parameter)	Block end
M08	Coolant ON	Block start
M09	Coolant OFF	Block end
M13	Spindle ON clockwise Coolant ON	Block start
M14	Spindle ON counterclockwise Coolant ON	Block start
M30	Same as M02	Block end

7.3 Miscellaneous Functions for Coordinate Data

Programming machine-referenced coordinates: M91/M92

Scale reference point
On the scale, a reference mark indicates the position of the scale reference point.

Machine datum
The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)

- Moving to machine-referenced positions (such as tool change positions)

- Setting the workpiece datum
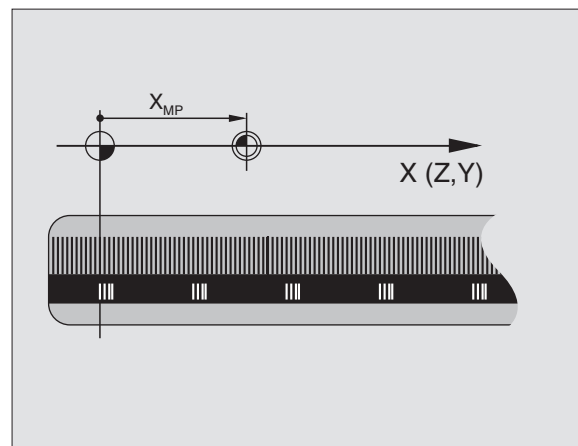
The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior
The TNC references coordinates to the workpiece datum (see "Datum setting").

Behavior with M91 — Machine datum
If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF (see section 1.4 "Status Displays").

Behavior with M92 — Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

M91 and M92 are not effective in a tilted working plane. If you program these M functions in a tilted plane, the TNC will display an error message.

Duration of effect
M91 and M92 are effective only in the blocks in which they are programmed with M91 or M92.

M91 and M92 become effective at the start of block.

Workpiece datum
If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes (see machine parameter 7295).

If datum setting is inhibited for all axes, the TNC no longer displays the soft key DATUM SET in the MANUAL OPERATION mode.

The figure at right shows coordinate systems with the machine datum and workpiece datum.
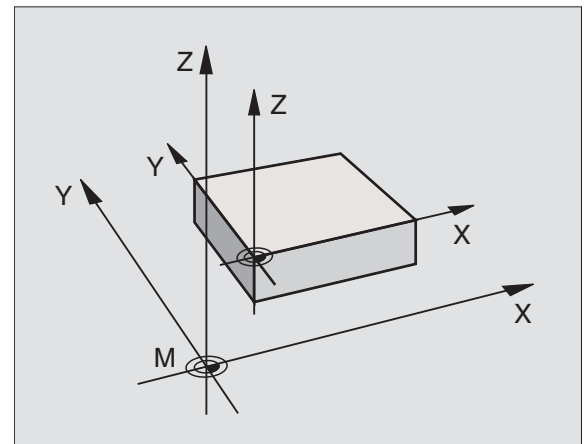
7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

In program blocks with radius compensation (RR/RL), the TNC automatically inserts a transition arc at outside corners.

Behavior with M90
The tool moves at corners with constant speed: This provides a smoother, more continuous surface and reduces machining time (see figure at lower right).

Example application: Surface consisting of a series of straight line segments.

Duration of effect
M90 is effective only in the blocks in which it is programmed with M90.

M90 becomes effective at the start of block. Operation with servo lag must be active.

☞ Independently of M90, you can use machine parameter MP7460 to set a limit value up to which the tool moves at constant path speed (effective with servo lag and feedforward control).
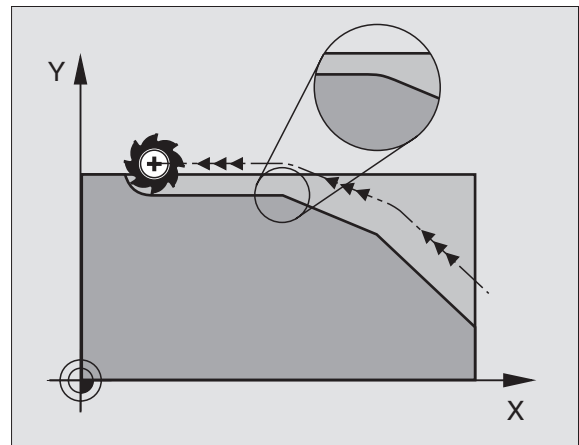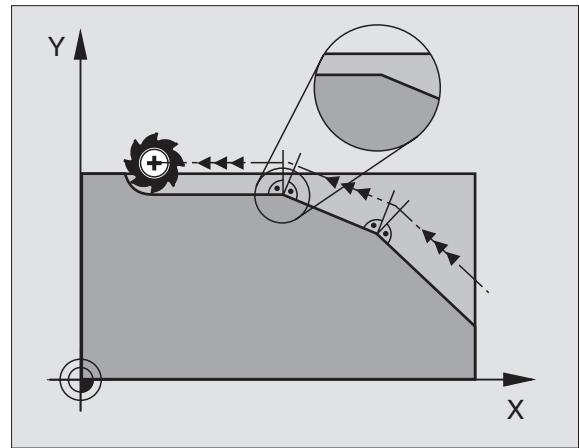
Insert rounding arc between straight lines: M112

Standard behavior

The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

In program blocks with radius compensation (RR/RL), the TNC automatically inserts a transition arc at outside corners.

> M112 is interfaced to the TNC and the machine by the machine tool builder. Observe the information in your machine manual.

Behavior with M112

The TNC inserts a rounding arc between two **uncompensated straight lines** (see figure at right). The size of the arc depends on:

- the permissible deviation from the programmed contour entered for T (if no deviation is entered, the TNC assumes the permissible deviation to be "infinite")

- the lengths of the two straight line segments between which the rounding arc is to be inserted

- the programmed feed rate (override setting 150%) and the circular acceleration (defined by the machine tool builder through machine parameters).

From these data, the TNC calculates the rounding arc with the smallest possible radius. If necessary, the contouring feed rate will automatically be reduced to comply with the programmed tolerance.

The permissible deviation T should be less than the programmed distance between points.

Tolerance angle A

If you have entered a tolerance angle A, the TNC takes the programmed feed rate into account for calculation of the rounding arc only if the direction angle is greater than the programmed tolerance angle.

Entering M112 in a positioning block

If you enter M112 in a positioning block, the TNC continues the dialog by asking you the permissible tolerance T and the tolerance angle A.

You can also define T through Q parameters — see Chapter 10 "Programming: Q Parameters."

Duration of effect

M112 is effective during operation with velocity feedforward as well as with servo lag.

M112 becomes effective at the start of block.

To cancel M112, enter M113.

Resulting NC block

```
L  X+123.723  Y+25.491  R0  F800  M112  T0.01  A10
```
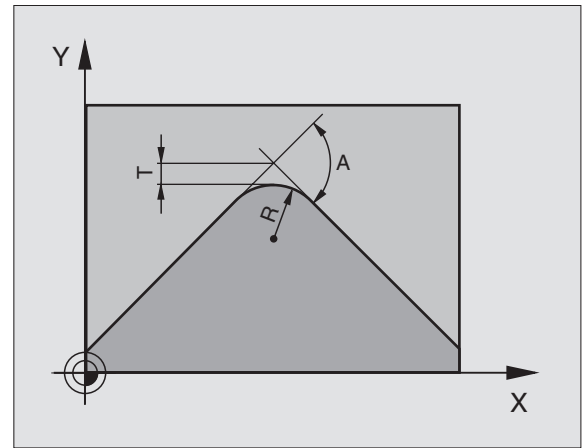
Ignore points for calculating the rounding arc with M112: M124

Standard behavior
All existing points are taken into account for calculating the rounding arc between two straight lines with M112.

Behavior with M124
Particularly when machining digitized 3-D surfaces, the distance between contour points at sections with sharp changes in direction may be too small to insert a rounding arc with M112. With the miscellaneous function M124, the TNC filters out such points. M124 is programmed by entering a minimum distance between points for the parameter T.

If the distance between two points is less than the programmed value, the TNC automatically **ignores** the second point and uses the **next** contour point for calculating the rounding arc.

Programming M124
If you enter M124 in a positioning block, the TNC continues the dialog for this block by asking you the minimum distance between points T.

You can also define T through Q parameters — see Chapter 10 "Programming: Q Parameters."

Duration of effect
M124 is effective at the start of the block and only if M112 is active. M124 and M112 are canceled by M113.

Resulting NC block

```
L  X+123.723  Y+25.491  R0  F800  M124  T0.01
```

Jolt reduction when changing the direction of traverse: M132

Standard behavior
Whenever the traverse direction changes, the change in the rate of acceleration creates a "jolt." Such jolts can leave small blemishes on the workpiece surface.

Behavior with M132
The TNC reduces the rate of change of acceleration during a change in direction. This feature smooths **all types of contour transitions** and results in a more regular workpiece surface. Jolt reduction can lead to slight contour deviations, which however in most cases are insignificant.

Duration of effect
M132 is effective at the start of the block.
M132 is canceled by M133

Machining small contour steps: M97

Standard behavior
The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour (see figure at upper right).

In such cases the TNC interrupts program run and generates the error message "TOOL RADIUS TOOL LARGE."

Behavior with M97
The TNC calculates the contour intersection of the contour elements — as at inside corners — and moves the tool over this point (see figure at lower right).

Program M97 in the same block as the outside corner.

Duration of effect
M97 is effective only in the blocks in which it is programmed with M97.

☞ A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.
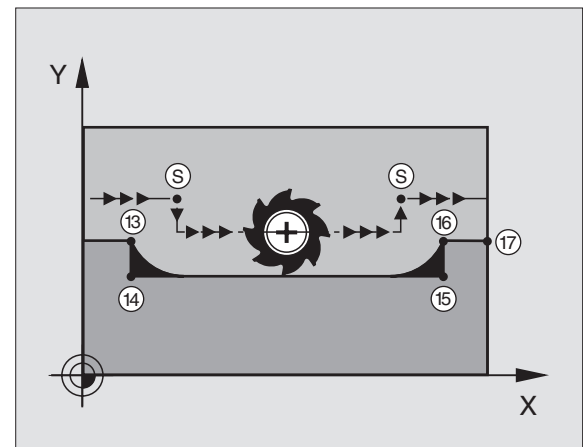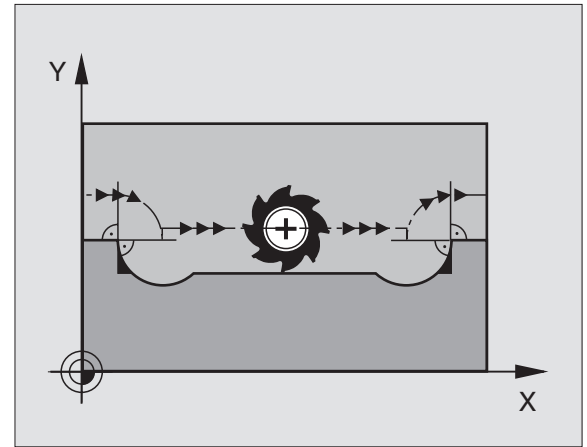
Resulting NC blocks

`5   TOOL DEF L ... R+20`	Large tool radius
`...`	
`13  L X ... Y ... R.. F .. M97`	Move to contour point 13
`14  L IY-0.5 .... R .. F..`	Machine small contour step 13 to 14
`15   L IX+100 ...`	Move to contour point 15
`16  L IY+0.5 ... R .. F.. M97`	Machine small contour step 15 to 16
`17  L X .. Y ...`	Move to contour point 17

Machining open contours: M98

Standard behavior
The TNC calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining (see figure at upper right).

Behavior with M98
With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined (see figure at lower right).

Duration of effect
M98 is effective only in the blocks in which it is programmed with M98.

M98 becomes effective at the end of block.

Resulting NC blocks
Move to the contour points 10, 11 and 12 in succession:

```
10  L X ... Y... RL F
11  L X... IY... M98
12  L IX+ ...
```

Feed rate factor for plunging movements: M103

Standard behavior
The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103
The TNC reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

$$FZMAX = FPROG \times F\%$$

Programming M103
If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

Duration of effect
M103 becomes effective at the start of block.
To cancel M103, program M103 once again **without a factor**.

Resulting NC blocks
The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
17 L X+20 Y+20 RL F500 M103 F20	500
18 L Y+50	500
19 L IZ–2.5	100
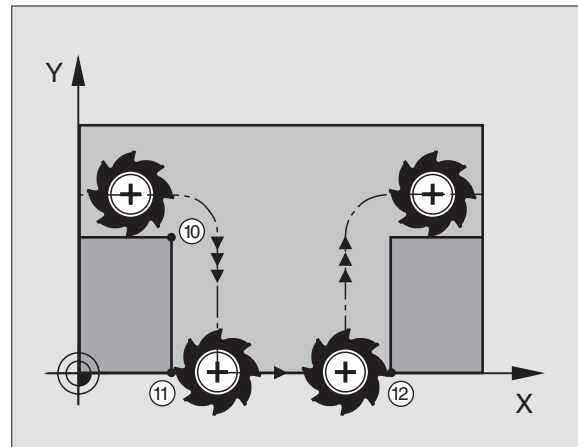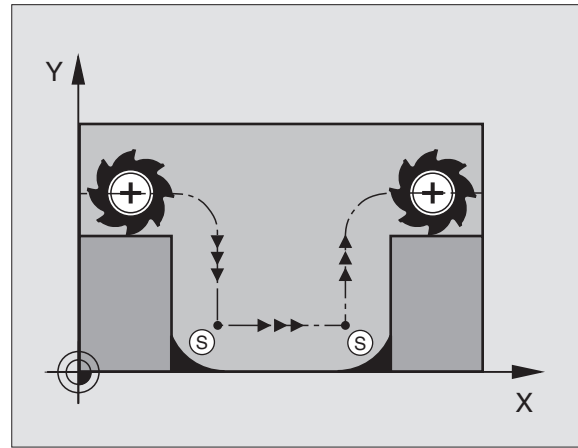
20 L IY+5 IZ–5	141
21 L IX+50	500
22 L Z+5	500

☞ M103 is activated with machine parameter 7440; see section 15.1 "General User Parameters."

Feed rate at circular arcs: M109/M110/M111

Standard behavior
The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109
The TNC adjusts the feed rate for circular arcs at inside and outside contours such that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110
The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

Duration of effect
M109 and M110 become effective at the start of block.
To cancel M109 or M110, enter M111.

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

Standard behavior
If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. Although you can use M97 to inhibit the error message (see "Machining small contour steps: M97"), this will result in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour — see figure at right.

Behavior with M120
The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Use LA (**L**ook **A**head) behind M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be — see figure at right.
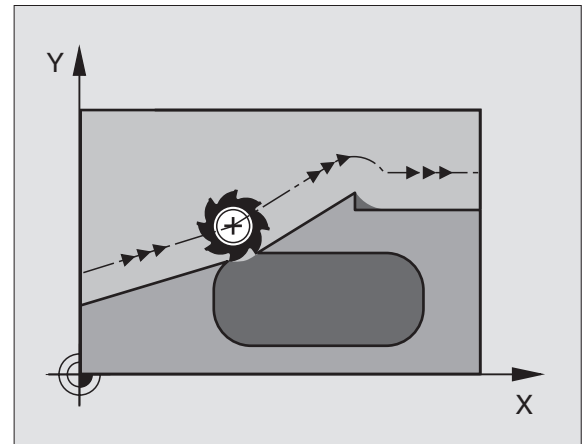
Programming M120

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Duration of effect

M120 must be located in an NC block that also contains radius compensation RL or RR. M120 is then effective from this block until

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA.

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS AT N.
- If you are using the path functions RND and CHF, the blocks before and after RND or CHF must contain only coordinates of the working plane.
- If you want to approach the contour on a tangential path, you must use the function APPR LCT. The block with APPR LCT must contain only coordinates of the working plane.
- If you want to approach the contour on a tangential path, use the function DEP LCT. The block with DEP LCT must contain only coordinates of the working plane.

Superimposing handwheel positioning during program run: M118

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. You can use this miscellaneous function by entering axis-specific values X, Y and Z (in mm) behind M118.

Programming M118

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons.

Duration of effect

Cancel handwheel positioning by programming M118 once again without X, Y and Z.

M118 becomes effective at the start of block.

Resulting NC block

You wish to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm of the programmed value:

```
L  X+0  Y+38.5  RL  F125  M118  X1  Y1
```

☞ M118 is always effective in the original working plane, even if the function for tilting the working plane is active.

M118 can also be used in the POSITIONING WITH MANUAL DATA INPUT mode.

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116

Standard behavior

The TNC interprets the programmed feed rate in a rotary axis in degrees per minute. The contouring feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

The TNC interprets the programmed feed rate in a rotary axis in mm/min. With this miscellaneous function, the TNC calculates the feed rate for each block at the **start** of the individual block. The feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Duration of effect

M116 is effective in the working plane and is automatically canceled at the end of program.

✎ The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior
If you reduce display of a rotary axis to a value less than 360°, the TNC will move the axis by the difference between the nominal and actual positions. The table at upper right shows examples.

Behavior with M126
With M126, the TNC will move the axis on the shorter path of traverse if you reduce display of a rotary axis to a value less than 360°. The table at lower right shows examples.

Duration of effect
M126 becomes effective at the start of block. To cancel M126, enter M127. At the end of program, M126 is automatically canceled.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior
The TNC moves the tool from the current angular value to the programmed angular value.

Example:
Current angular value: 538°
Programmed angular value: 180°
Actual path of traverse: –358°

Behavior with M94
At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. To have the TNC reduce the display for a specific rotary axis only, enter the axis after M94.

Resulting NC blocks
To reduce display of all active rotary axes:

```
L  M94
```

To reduce display of the C axis only:

```
L  M94  C
```

To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
L  C+180  FMAX  M94
```

Duration of effect
M94 is effective only in the block in which M94 is programmed.

M94 becomes effective at the start of block.

Standard behavior of the TNC

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Automatic compensation of machine geometry when working with tilted axes: M114

Standard behavior

The TNC moves the tool to the positions given in the part program. The offset of the tool for positioning in tilted axes must be calculated by a postprocessor.

Behavior with M114

The TNC compensates the tool offset by a 3-D length compensation. The radius compensation must be calculated by a CAD system or by a postprocessor. A programmed radius compensation RL/RR results in the error message "ILLEGAL NC BLOCK."

The figure at right illustrates the offset of the tool datum while swiveling.

If you write the NC program with a postprocessor, the machine geometry does not have to be calculated.

If the tool length compensation is calculated by the TNC, the programmed feed rate refers to the point of the tool. Otherwise it refers to the tool datum.

☞ If you machine tool is equipped with a swivel head that can be tilted under program control, you can interrupt program run and change the position of the tilted axis, for example with the handwheel.

With the RESTORE POS AT N function, you can resume program run at the block at which the part program was interrupted. The TNC automatically calculates the new position of the tilted axis.

Duration of effect

M114 becomes effective at the start of block, M115 at the end of block.

To cancel M114, enter M115. At the end of program, M114 is automatically canceled.

✍ The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.
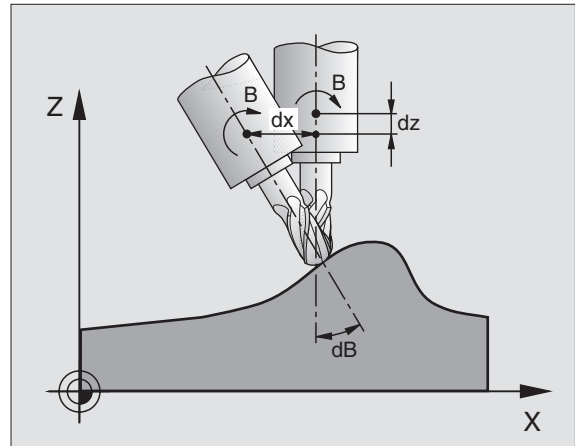
7.6 Miscellaneous Functions for Laser Cutting Machines

The TNC can control the cutting efficiency of a laser by transferring voltage values through the S-analog output. You can influence laser efficiency during program run through the miscellaneous functions M200 to M204.

Entering miscellaneous functions for laser cutting machines
If you enter an M function for laser cutting machines in a positioning block, the TNC continues the dialog by asking you the required parameters for the programmed function.

All miscellaneous functions for laser cutting machines become effective at the start of block.

Output the programmed voltage directly: M200

The TNC outputs the value programmed after M200 as the voltage V.

Input range: 0 to 9.999 V

Duration of effect
M200 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of distance: M201

M201 outputs the voltage in dependence on the distance to be covered. The TNC increases or decreases the current voltage linearly to the value programmed for V.

Input range: 0 to 9.999 V

Duration of effect
M201 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of speed: M202

The TNC outputs the voltage as a function of speed. In the machine parameters, the machine tool builder defines up to three characteristic curves FNR in which specific feed rates are assigned to specific voltages. Use miscellaneous function M202 to select the curve FNR from which the TNC is to determine the output voltage.

Input range: 1 to 3

Duration of effect
M202 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent ramp): M203

The TNC outputs the voltage V as a function of the time TIME. The TNC increases or decreases the current voltage linearly to the value programmed for V within the time programmed for TIME.

Input range
Voltage V: 0 to 9.999 Volt
TIME: 0 to 1.999 seconds

Duration of effect
M203 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent pulse): M204

The TNC outputs a programmed voltage as a pulse with a programmed duration TIME.

Input range
Voltage V: 0 to 9.999 Volt
TIME: 0 to 1.999 seconds

Duration of effect
M204 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

8

Programming:

Cycles

8.1 General Overview of Cycles

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles. The table at right shows the different groups of cycles that are available on your TNC.

Fixed cycles from Cycle 200 on use Q parameters as the transfer parameters for the input values. Parameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the setup clearance, Q202 the pecking depth, etc.

To define a cycle

▶ The soft-key row shows the available groups of cycles

▶ Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles

▶ Select the desired cycle, for example PECKING. The TNC initiates the programming dialog and asks all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted

▶ Enter all parameters asked by the TNC and conclude each entry with the ENT key

▶ The TNC terminates the dialog when all required data have been entered

Group of cycles	Soft key
Cycles for pecking, reaming, boring, tapping and thread cutting	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/ISLANDS
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
SL (**S**ubcontour **L**ist) cycles, which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL II
Cycles for multipass milling of flat or twisted surfaces	MULTIPASS MILLING
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call and oriented spindle stop	SPECIAL CYCLES

Resulting NC blocks

CYCL DEF 1.0	PECKING
CYCL DEF 1.1	SET UP 2
CYCL DEF 1.2	DEPTH –30
CYCL DEF 1.3	PECKG 5
CYCL DEF 1.4	DWELL 1
CYCL DEF 1.5	F150

Cycle call

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Cycles for circular and linear hole patterns

- SL cycle CONTOUR GEOMETRY

- SL cycle CONTOUR DATA

- Coordinate transformation cycles

- DWELL TIME cycle

All other cycles are called as described below.

If the TNC is to execute the cycle once after the last programmed block, program the cycle call with the miscellaneous function M99 or with CYCL CALL:

| CYCL CALL | ▶ To program the cycle call, press the CYCL CALL key. |

▶ Enter a miscellaneous function, for example for coolant supply.

If the TNC is to execute the cycle automatically after every positioning block, program the cycle call with M89 (depending on machine parameter 7440).

To cancel M89, enter

- M99 or

- CYCL CALL or

- CYCL DEF

8.2 Drilling Cycles

The TNC offers 8 cycles for all types of drilling operations:

Cycle	Soft key
1 PECKING Without automatic pre-positioning	
200 DRILLING With automatic pre-positioning and 2nd setup clearance	
201 REAMING With automatic pre-positioning and 2nd setup clearance	
202 BORING With automatic pre-positioning and 2nd setup clearance	
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd setup clearance, chip breaking, and decrement	
2 TAPPING With a floating tap holder	
17 RIGID TAPPING Without a floating tap holder	
18 THREAD CUTTING	

PECKING (Cycle 1)

1 The tool drills from the current position to the first PECKING DEPTH at the programmed FEED RATE F.

2 When it reaches the first pecking depth, the tool retracts in rapid traverse FMAX to the starting position and advances again to the first PECKING DEPTH minus the advanced stop distance t.

3 The advanced stop distance is automatically calculated by the control:

- At a total hole depth of up to 30 mm: t = 0.6 mm

- At a total hole depth exceeding 30 mm: t = hole depth / 50

Maximum advanced stop distance: 7 mm

4 The tool then advances with another infeed at the programmed FEED RATE F.

5 The TNC repeats this process (1 to 4) until the programmed TOTAL HOLE DEPTH is reached.

6 After a dwell time at the hole bottom, the tool is returned to the starting position in rapid traverse FMAX for chip breaking.
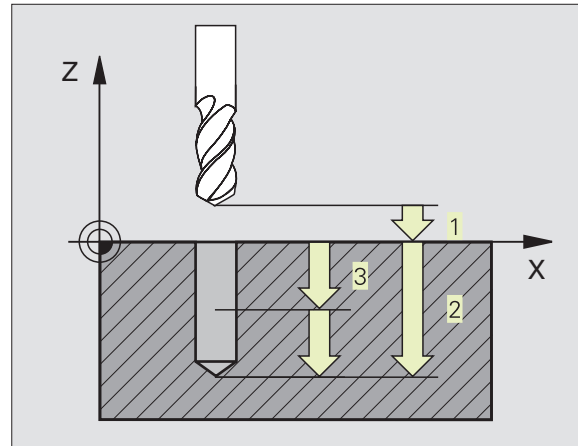
▶ SET-UP CLEARANCE **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface

▶ TOTAL HOLE DEPTH **2** (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)

▶ PECKING DEPTH **3** (incremental value): Infeed per cut. The tool will drill to the TOTAL HOLE DEPTH in one operation if:

- the PECKING DEPTH equals the TOTAL HOLE DEPTH

- the PECKING DEPTH is greater than the TOTAL HOLE DEPTH

The TOTAL HOLE DEPTH does not have to be a multiple of the PECKING DEPTH.

▶ DWELL TIME IN SECONDS: Amount of time the tool remains at the total hole depth for chip breaking

▶ FEED RATE F: Traversing speed of the tool during drilling in mm/min

DRILLING (Cycle 200)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the SET-UP CLEARANCE above the workpiece surface.

2 The tool drills to the first PECKING DEPTH at the programmed FEED RATE F.

3 The tool is retracted to SET-UP CLEARANCE in FMAX, remains there — if programmed — for the entered dwell time, and advances again in FMAX to 0.2 mm above the first PECKING DEPTH.

4 The tool then advances with another infeed at the programmed FEED RATE F.

5 The TNC repeats this process (1 to 4) until the programmed TOTAL HOLE DEPTH is reached.

6 At the hole bottom, the tool is retraced to SET-UP CLEARANCE or — if programmed — to the 2ND SET-UP CLEARANCE in rapid traverse FMAX.
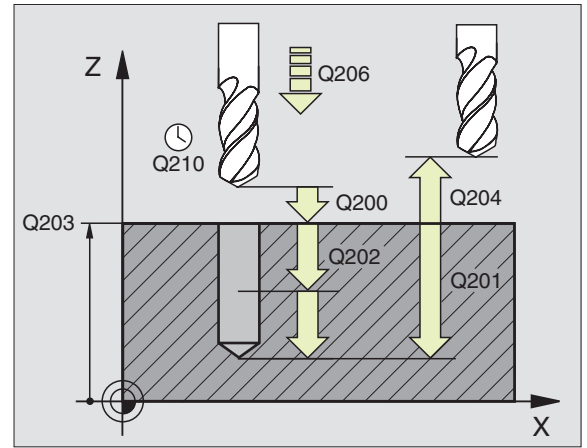
☞ **Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the cycle parameter DEPTH determines the working direction.

Cycle 200 is an abbreviated version of the UNIVERSAL DRILLING Cycle 203, but still provides the full range of functions of Cycle 203. The additional parameters can be assigned after the cycle definition:
Q parameter function FN0: ASSIGN

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool during drilling in mm/min

▶ PECKING DEPTH Q202 (incremental value): Infeed per cut. The tool will drill to the DEPTH in one operation if:

■ the PECKING DEPTH equals the DEPTH

■ the PECKING DEPTH is greater than the DEPTH

The DEPTH does not have to be a multiple of the PECKING DEPTH.

▶ DWELL TIME AT TOP Q210: Time in seconds that the tool remains at SET-UP CLEARANCE after having been retracted from the hole for chip release.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

REAMING (Cycle 201)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed SET-UP CLEARANCE above the workpiece surface.

2 The tool reams to the entered DEPTH at the programmed FEED RATE F.

3 If programmed, the tool remains at the hole bottom for the entered dwell time.

4 The tool then retracts to SET-UP CLEARANCE at the FEED RATE F, and from there — if programmed — to the 2ND SET-UP CLEAR-ANCE in FMAX.

☞ **Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSA-TION R0.

The algebraic sign for the cycle parameter DEPTH determines the working direction.
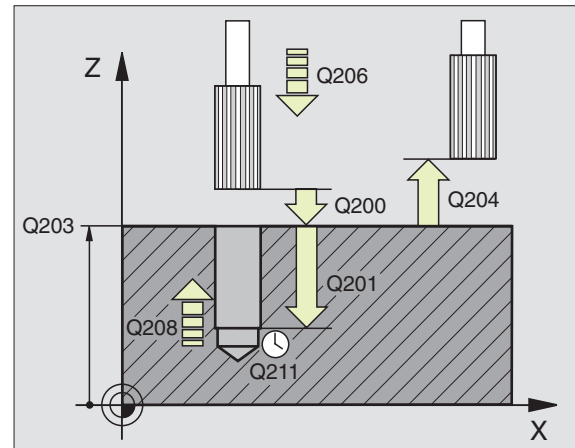
▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of hole

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool during reaming in mm/min

▶ DWELL TIME AT DEPTH Q211: Time in seconds that the tool remains at the hole bottom

▶ RETRACTION FEED RATE Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the REAMING FEED RATE.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

BORING (Cycle 202)

> Machine and control must be specially prepared by the machine tool builder to enable Cycle 202.

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to SET-UP CLEARANCE above the workpiece surface.

2 The tool drills to the programmed DEPTH at the FEED RATE FOR PLUNGING.

3 If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.

4 The TNC then orients the spindle to the 0° position with an oriented spindle stop.

5 If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).

6 The tool then retracts to SET-UP CLEARANCE at the RETRACTION RATE F, and from there — if programmed — to the 2ND SET-UP CLEARANCE in FMAX.

> ☞ **Before programming, note the following:**
>
> Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.
>
> The algebraic sign for the cycle parameter DEPTH determines the working direction.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of hole

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool during boring in mm/min

▶ DWELL TIME AT DEPTH Q211: Time in seconds that the tool remains at the hole bottom

▶ RETRACTION FEED RATE Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at FEED RATE FOR PLUNGING.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
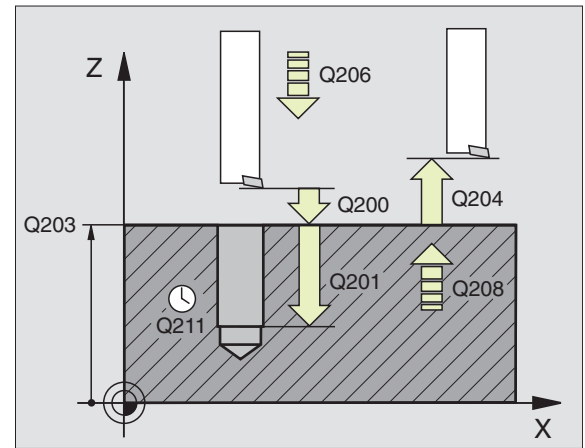
▶ DISENGAGING DIRECTION (0/1/2/3/4) Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

0: Do not retract tool

1: Retract tool in the negative main axis direction

2: Retract tool in the negative secondary axis direction

3: Retract tool in the positive main axis direction

4: Retract tool in the positive secondary axis direction

☞ **Danger of collision!**

Check the position of the tool tip when you program a spindle orientation to 0° (for example, in the POSITIONING WITH MANUAL DATA INPUT mode of operation). Align the tool tip so that it is parallel to a coordinate axis. Select a DISENGAGING DIRECTION in which the tool moves away from the edge of the hole.

UNIVERSAL DRILLING (Cycle 203)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed SET-UP CLEARANCE above the workpiece surface.

2 The tool drills to the first PECKING DEPTH at the programmed FEED RATE F.

3 If you have programmed chip breaking, the tool then retracts by 0.2 mm. If you are working without chip breaking, the tool retracts at the RETRACTION FEED RATE to SET-UP CLEARANCE, remains there — if programmed — for the entered dwell time, and advances again in FMAX to 0.2 mm above the first PECKING DEPTH.

4 The tool then advances with another infeed at the programmed FEED RATE. If programmed, the PECKING DEPTH is decreased after each infeed by the DECREMENT.

5 The TNC repeats this process (2 to 4) until the programmed TOTAL HOLE DEPTH is reached.

6 The tool remains at the hole bottom — if programmed — for the entered DWELL TIME to cut free, and then retracts to SET-UP CLEARANCE at the RETRACTION FEED RATE. If you have entered a 2ND SET-UP CLEARANCE, the tool subsequently moves to that position in FMAX.

☞ **Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the cycle parameter DEPTH determines the working direction.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool during drilling in mm/min

▶ PECKING DEPTH Q202 (incremental value): Infeed per cut. The tool will drill to the DEPTH in one operation if:
  ■ the PECKING DEPTH equals the DEPTH
  ■ the PECKING DEPTH is greater than the DEPTH

  The DEPTH does not have to be a multiple of the PECKING DEPTH.

▶ DWELL TIME AT TOP Q210: Time in seconds that the tool remains at SET-UP CLEARANCE after having been retracted from the hole for chip breaking.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

▶ DECREMENT Q212 (incremental value): Value by which the TNC decreases the PECKING DEPTH after each infeed.

▶ NR OF BREAKS BEFORE RETRACTING Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip release. For chip breaking, the TNC retracts the tool each time by 0.2 mm.

▶ MINIMUM PECKING DEPTH Q205 (incremental value): If you have entered a decrement, the TNC limits the PECKING DEPTH to the value entered with Q205.

▶ DWELL TIME AT DEPTH Q211: Time in seconds that the tool remains at the hole bottom

▶ RETRACTION FEED RATE Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts in FMAX.

TAPPING with a floating tap holder (Cycle 2)

1 The thread is cut in one pass.

2 Once the tool has reached the TOTAL HOLE DEPTH, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the DWELL TIME.

3 At the starting position, the direction of spindle rotation reverses once again.
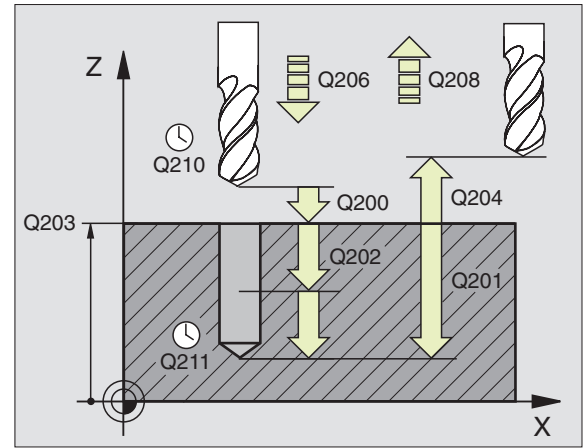
☞ **Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter TOTAL HOLE DEPTH determines the working direction.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is active only within a limited range (preset by the machine tool builder).

For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

▶ SET-UP CLEARANCE **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 x thread pitch

▶ TOTAL HOLE DEPTH **2** (thread length, incremental value): Distance between workpiece surface and end of thread

▶ DWELL TIME IN SECONDS: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.

▶ FEED RATE F: Traversing speed of the tool during tapping

The feed rate is calculated as follows: F = S x p
where   F is the feed rate in mm/min,
        S is the spindle speed in rpm, and
        p is the thread pitch in mm

Retracting after a program interruption
If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.
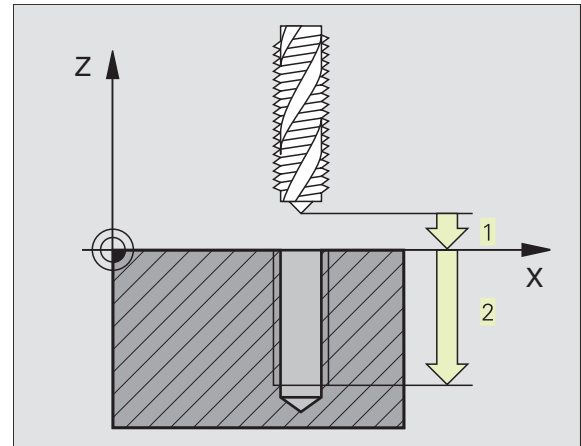
RIGID TAPPING (Cycle 17)

⌖ Machine and control must be specially prepared by the machine tool builder to enable rigid tapping.

The TNC cuts the thread without a floating tap holder in one or more passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder:

- Higher machining speeds possible

- Repeated tapping of the same thread is possible; repetitions are enabled via spindle orientation to the 0° position during cycle call (depending on a machine parameter).

- Increased traverse range of the spindle axis due to absence of a floating tap holder.

☞ **Before programming, note the following:**

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSA-TION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter TOTAL HOLE DEPTH determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

▶ SET-UP CLEARANCE **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface

▶ TOTAL HOLE DEPTH **2** (incremental value): Distance between workpiece surface (beginning of thread) and end of thread

▶ PITCH **3** :
Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
– = left-hand thread

Retracting after a program interruption
If you interrupt program run during tapping with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.
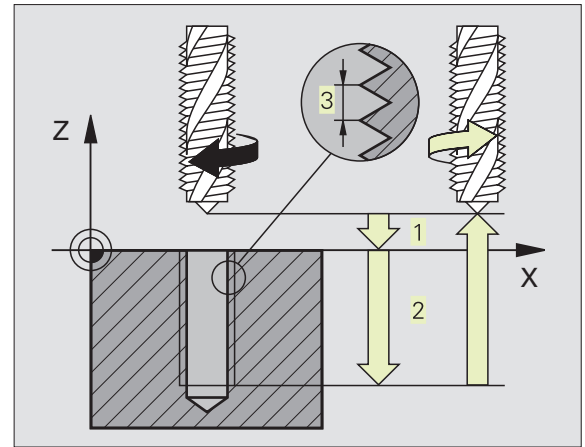
THREAD CUTTING (Cycle 18)

> Machine and control must be specially prepared by the machine tool builder to enable thread cutting.

Cycle 18 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered DEPTH. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient way to do this is by using OEM cycles. The machine tool builder can give you further information.

Before programming, note the following:

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted.

The feed rate override knob is disabled.

The TNC automatically activates and deactivates spindle rotation. Do not program M3 or M4 before cycle call.

▶ TOTAL HOLE DEPTH 1 : Distance between current tool position and end of thread.

The algebraic sign for the TOTAL HOLE DEPTH determines the working direction (a negative value means a negative working direction in the tool axis)

▶ PITCH 2 :
Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread (M3 with negative DEPTH)
– = left-hand thread (M4 with negative DEPTH)
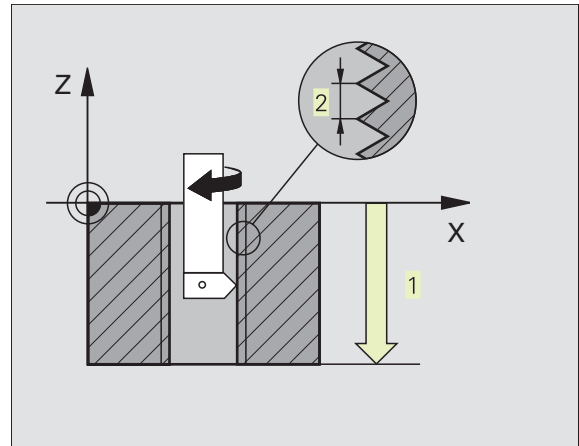
Example: Drilling cycles
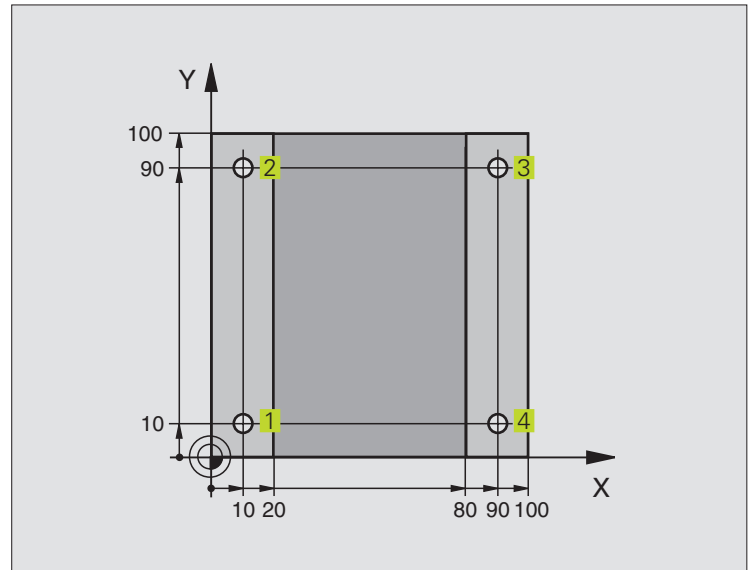
0	BEGIN PGM C200 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+3	Define tool
4	TOOL CALL 1 Z S4500	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	CYCL DEF 200 DRILLING	Define cycle
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q202=5 ;PECKING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=-10 ;SURFACE COORDINATE	
	Q204=20 ;2ND SET-UP CLEARANCE	
7	L X+10 Y+10 R0 F MAX M3	Approach hole 1, spindle ON
8	CYCL CALL	Call cycle
9	L Y+90 R0 F MAX M99	Approach hole 2, call cycle
10	L X+90 R0 F MAX M99	Approach hole 3, call cycle
11	L Y+10 R0 F MAX M99	Approach hole 4, call cycle
12	L Z+250 R0 F MAX M2	Retract tool, end of program
13	END PGM C200 MM	

Example: Drilling cycles

Process
- Program the drilling cycle in the main program
- Program the machining operation in a subprogram (see section 9 "Programming: Subprograms and Program Section Repeats")
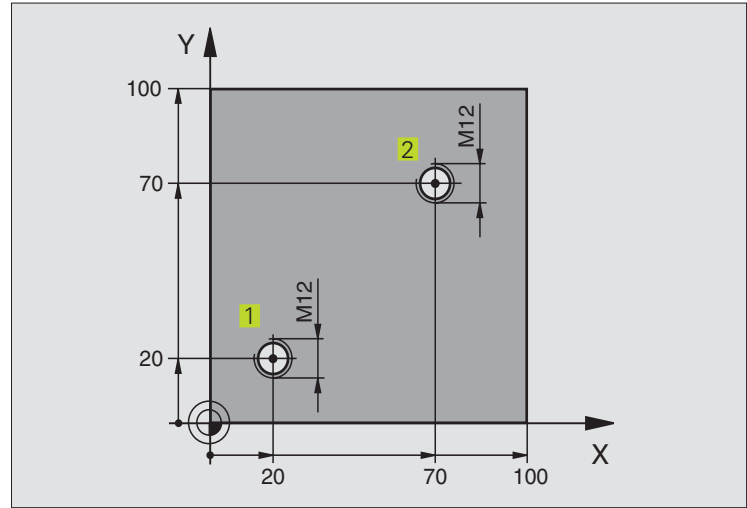
0	BEGIN PGM C18 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+6	Define tool
4	TOOL CALL 1 Z S100	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	CYCL DEF 18.0 THREAD CUTTING	Define THREAD CUTTING cycle
7	CYCL DEF 18.1 DEPTH +30	
8	CYCL DEF 18.2 PITCH -1.75	
9	L X+20 Y+20 R0 F MAX	Approach hole 1
10	CALL LBL 1	Call subprogram 1
11	L X+70 Y+70 R0 F MAX	Approach hole 2
12	CALL LBL 1	Call subprogram 1
13	L Z+250 R0 F MAX M2	Retract tool, end of main program
14	LBL 1	Subprogram 1: Thread cutting
15	CYCL DEF 13.0 ORIENTATION	Orient spindle (makes it possible to cut repeatedly)
16	CYCL DEF 13.1 ANGLE 0	
17	L IX-2 R0 F1000	Tool offset to prevent collision during tool infeed (dependent on core diameter and tool)
18	L Z+5 R0 F MAX	Pre-position in rapid traverse
19	L Z-30 R0 F1000	Move to starting depth
20	L IX+2	Reset the tool to hole center
21	CYCL CALL	Call Cycle 18
22	L Z+5 R0 F MAX	Retract tool
23	LBL 0	End of subprogram 1
24	END PGM C18 MM	

8.3 Cycle for Milling Pockets, Studs and Slots

Cycle	Soft key
4 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning	
212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd setup clearance	
213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd setup clearance	
5 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning	
214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning and 2nd setup clearance	
215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning and 2nd setup clearance	
3 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical downfeed	
210 SLOT WITH RECIPROCATING PLUNGE-CUT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	
211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	

POCKET MILLING (Cycle 4)

1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first PECKING DEPTH.

2 The tool subsequently follows the programmed path at the FEED RATE F — see figure at right. The cutter begins milling in the positive axis direction of the longer side. On square pockets, the cutter always starts in the positive Y direction.

3 This process (1 to 3) is repeated until the DEPTH is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.

☞ **Before programming, note the following:**

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter DEPTH determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

The following condition must be met for the SECOND SIDE LENGTH:
2nd SIDE LENGTH greater than [(2 x ROUNDING-OFF RADIUS) + stepover factor k].
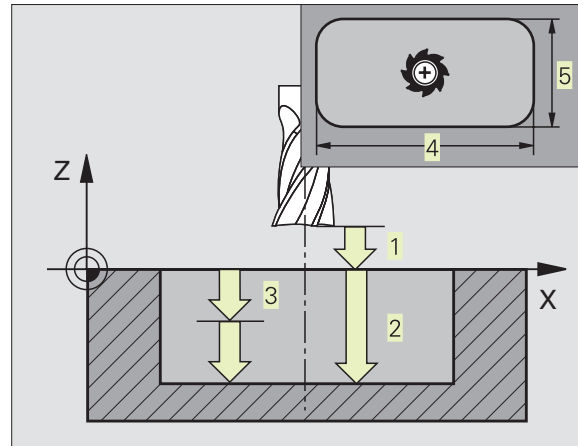
▶ SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface

▶ MILLING DEPTH 2 (incremental value): Distance between workpiece surface and bottom of pocket

▶ PECKING DEPTH 3 (incremental value): Infeed per cut. The tool will advance to the DEPTH in one operation if:
- the PECKING DEPTH equals the DEPTH
- the PECKING DEPTH is greater than the DEPTH

▶ FEED RATE FOR PECKING: Traversing speed of the tool during penetration

▶ FIRST SIDE LENGTH 4: Pocket length, parallel to the main axis of the working plane

▶ SECOND SIDE LENGTH 5: Pocket width

▶ FEED RATE F: Traversing speed of the tool in the working plane

▶ DIRECTION OF THE MILLING PATH
DR **+** : climb milling with M3
DR **–** : up-cut milling with M3

▶ ROUNDING-OFF RADIUS: RADIUS for the pocket corners. If RADIUS = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

Calculations:
Stepover factor k = K x R

where
K   is the overlap factor, preset in machine parameter 7430, and
R   is the cutter radius

POCKET FINISHING (Cycle 212)

1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEAR-ANCE, and subsequently to the center of the pocket.

2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the finishing allowance and tool radius into account for calculating the starting point.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PECKING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

6 This process (2 to 5) is repeated until the programmed DEPTH is reached.

7 At the end of the cycle, the TNC retracts the tool in rapid traverse to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and finally to the center of the pocket (end position = starting position).

☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

If you want to finish the pocket from solid metal, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of pocket

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool in mm/min when moving to DEPTH. If you are plunge-cutting into the material, enter a low value; if you have already cleared the pocket, enter a higher feed rate.

▶ PECKING DEPTH Q202 (incremental value): Infeed per cut; enter a value greater than 0.

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool during milling in mm/min.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the pocket in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the pocket in the secondary axis of the working plane

▶ FIRST SIDE LENGTH Q218 (incremental value): Pocket length, parallel to the main axis of the working plane

▶ SECOND SIDE LENGTH Q219 (incremental value): Pocket length, parallel to the secondary axis of the working plane

▶ CORNER RADIUS Q220: Radius of the pocket corner

▶ ALLOWANCE IN 1ST AXIS Q221 (incremental value): Allowance in the main axis of the working plane referenced to the length of the pocket
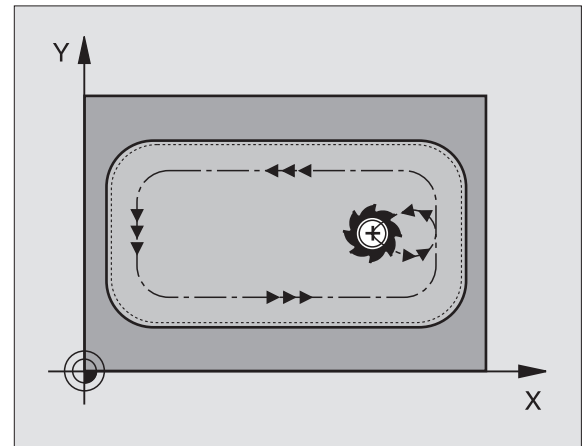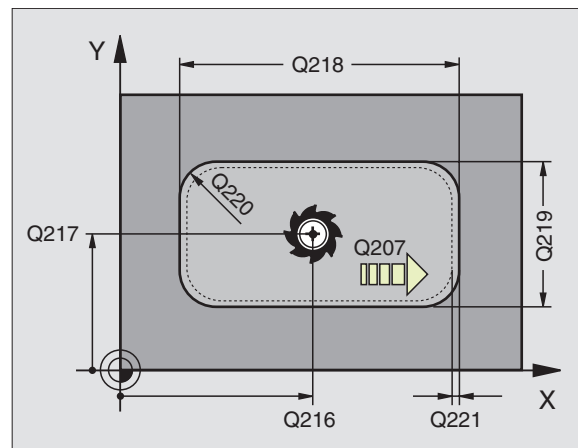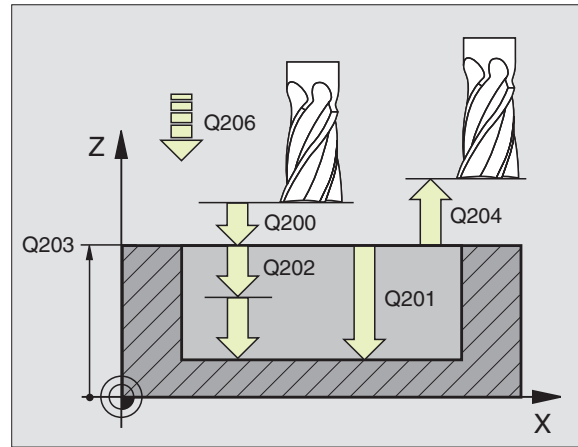
STUD FINISHING (Cycle 213)

1 The TNC moves the tool in the tool axis to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and subsequently to the center of the stud.

2 From the stud center, the tool moves in the working plane to the starting point for machining. The TNC takes the finishing allowance and tool radius into account for calculating the starting point.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PECKING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

6 This process (2 to 5) is repeated until the programmed DEPTH is reached.

7 At the end of the cycle, the TNC retracts the tool in FMAX to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and finally to the center of the stud (end position = starting position).

☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

If you want to mill a stud from solid metal, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of stud

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool in mm/min when moving to DEPTH. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.

▶ PECKING DEPTH Q202 (incremental value): Infeed per cut; enter a value greater than 0.

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool during milling in mm/min.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the stud in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the stud in the secondary axis of the working plane

▶ FIRST SIDE LENGTH Q218 (incremental value): Stud length, parallel to the main axis of the working plane

▶ SECOND SIDE LENGTH Q219 (incremental value): Stud length, parallel to the secondary axis of the working plane

▶ CORNER RADIUS Q220: Radius of the stud corner

▶ ALLOWANCE IN 1ST AXIS Q221 (incremental value): Allowance in the main axis of the working plane referenced to the length of the stud
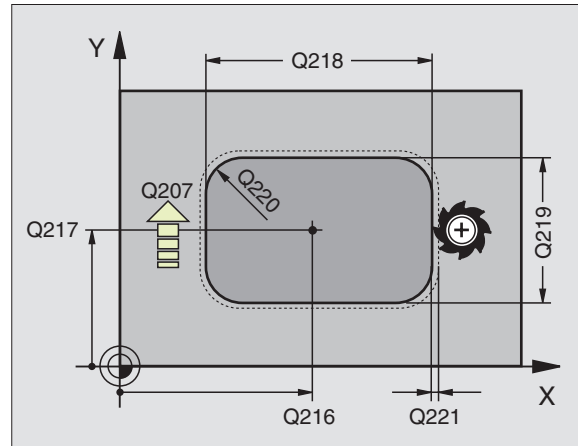
CIRCULAR POCKET MILLING (Cycle 5)

1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first PECKING DEPTH.

2 The tool subsequently follows a spiral path at the FEED RATE F — see figure at right. For calculating the stepover factor k, see Cycle 4 POCKET MILLING.

3 This process is repeated until the DEPTH is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.

☞ **Before programming, note the following:**

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter DEPTH determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

▶ SET-UP CLEARANCE **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface

▶ MILLING DEPTH **2** (incremental value): Distance between workpiece surface and bottom of pocket

▶ PECKING DEPTH **3** (incremental value): Infeed per cut. The tool will advance to the DEPTH in one operation if:
  ■ the PECKING DEPTH equals the DEPTH
  ■ the PECKING DEPTH is greater than the DEPTH

▶ FEED RATE FOR PECKING: Traversing speed of the tool during penetration

▶ CIRCLE RADIUS: Radius of the circular pocket

▶ FEED RATE F: Traversing speed of the tool in the working plane

▶ DIRECTION OF THE MILLING PATH
  DR **+** : climb milling with M3
  DR **–** : up-cut milling with M3
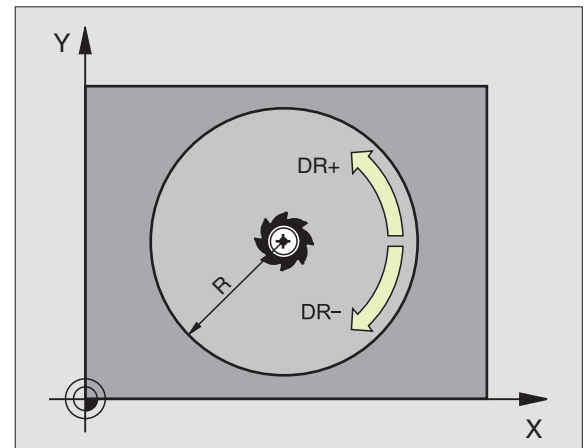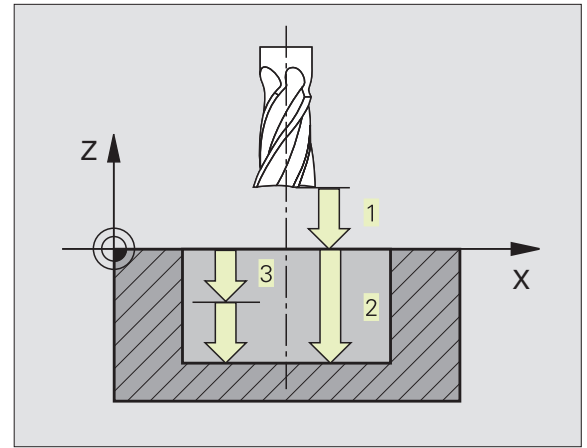
CIRCULAR POCKET FINISHING (Cycle 214)

1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEAR-ANCE, and subsequently to the center of the pocket.

2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PECKING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

6 This process (2 to 5) is repeated until the programmed DEPTH is reached.

7 At the end of the cycle, the TNC retracts the tool in FMAX to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and finally to the center of the pocket (end position = starting position).
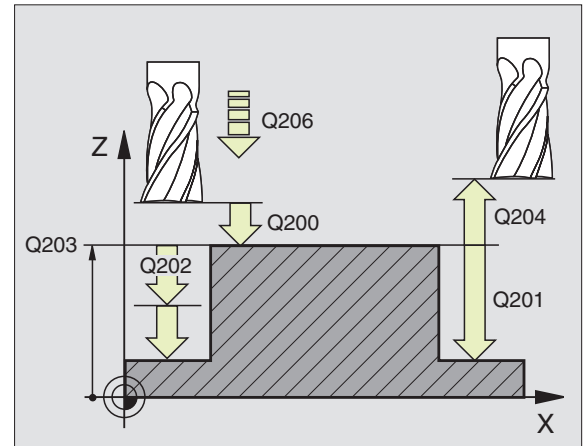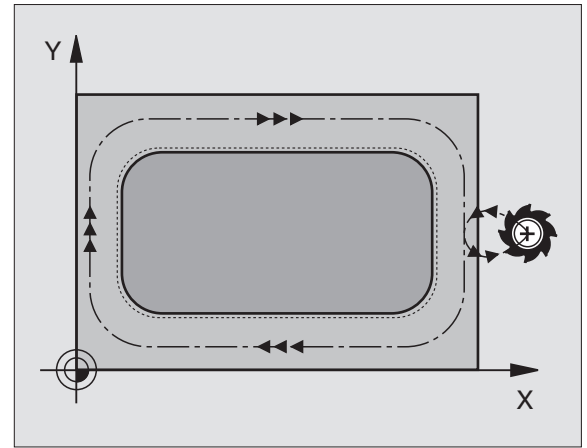
☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

If you want to finish the pocket from solid metal, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

▸ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▸ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of pocket

▸ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool in mm/min when moving to DEPTH. If you are plunge-cutting into the material, enter a low value; if you have already cleared the pocket, enter a higher feed rate.

▸ PECKING DEPTH Q202 (incremental value): Infeed per cut

▸ FEED RATE FOR MILLING Q207: Traversing speed of the tool during milling in mm/min.
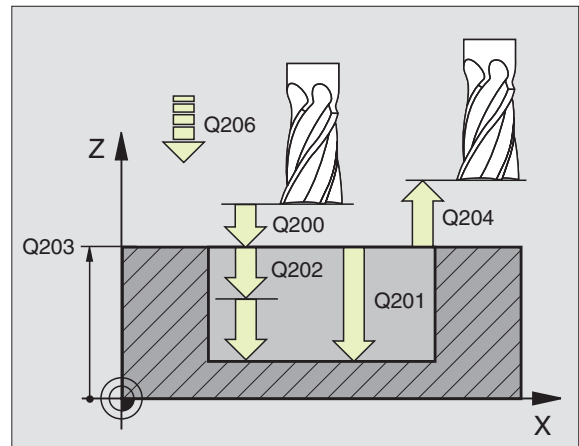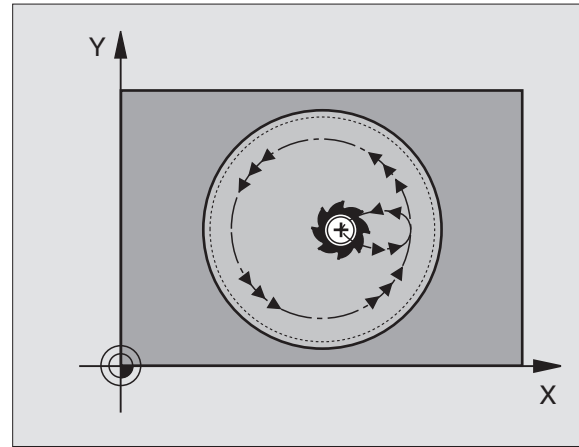
▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the pocket in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the pocket in the secondary axis of the working plane

▶ WORKPIECE BLANK DIA. Q222: Diameter of the premachined pocket. Enter the workpiece blank diameter to be less than the diameter of the finished part.

▶ FINISHED PART DIA. Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter.
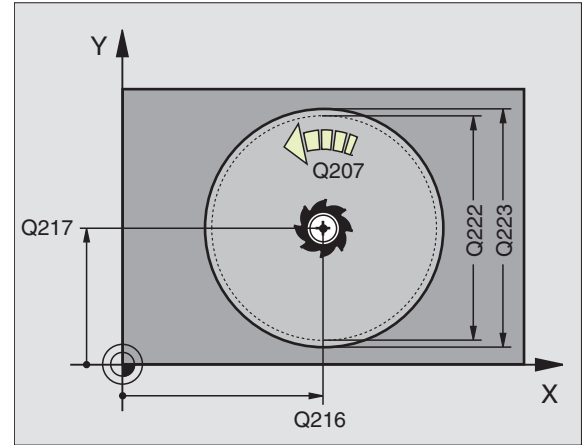
CIRCULAR STUD FINISHING (Cycle 215)

1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEAR-ANCE, and subsequently to the center of the stud.

2 From the stud center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PECKING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

6 This process (2 to 5) is repeated until the programmed DEPTH is reached.

7 At the end of the cycle, the TNC retracts the tool in FMAX to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and finally to the center of the stud (end position = starting position).

☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

If you want to mill a stud from solid, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of stud

▶ FEED RATE Q206 FOR PLUNGING: Traversing speed of the tool in mm/min when moving to DEPTH. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.

▶ PECKING DEPTH Q202 (incremental value): Infeed per cut; enter a value greater than 0.

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool during milling in mm/min.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the stud in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the stud in the secondary axis of the working plane

▶ WORKPIECE BLANK DIAMETER Q222: Diameter of the premachined stud. Enter the workpiece blank diameter to be greater than the diameter of the finished part.

▶ DIAMETER OF FINISHED PART Q223: Diameter of the finished stud. Enter the diameter of the finished part to be less than the workpiece blank diameter.
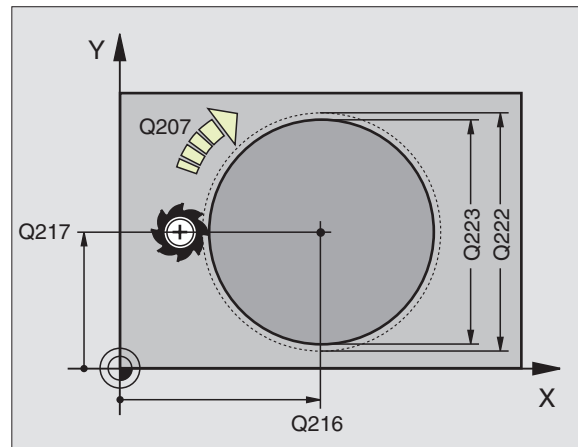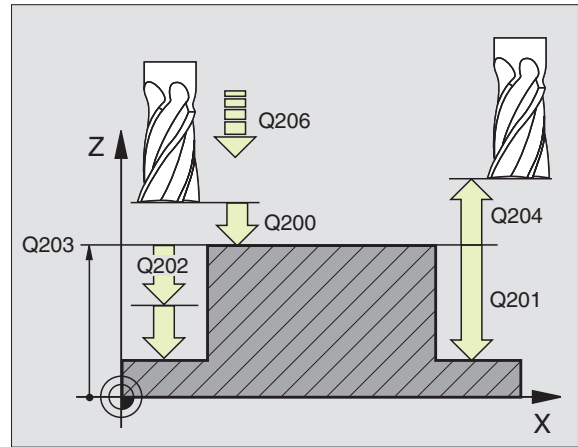
SLOT MILLING (Cycle 3)

Roughing process

1 The tool penetrates the workpiece from the starting position and mills in the longitudinal direction of the slot.

2 After downfeed at the end of the slot, milling is performed in the opposite direction.

This process is repeated until the programmed MILLING DEPTH is reached.

Finishing process

3 The TNC advances the tool at the slot bottom on a tangential arc to the outside contour. The tool subsequently climb mills the contour (with M3).

4 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE.

If the number of infeeds was odd, the tool returns to the starting position at the level of the SET-UP CLEARANCE.

☞ **Before programming, note the following:**

Program a positioning block for the starting point in the working plane — to the center of the slot (SECOND SIDE LENGTH) and, within the slot, offset by the tool radius — with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter DEPTH determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

The cutter diameter must be not be larger than the SLOT WIDTH and not smaller than half the SLOT WIDTH.

▶ SET-UP CLEARANCE **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface

▶ MILLING DEPTH **2** (incremental value): Distance between workpiece surface and bottom of pocket

▶ PECKING DEPTH **3** (incremental value): Infeed per cut. The tool will advance to the DEPTH in one operation if:
  ■ the PECKING DEPTH equals the DEPTH
  ■ the PECKING DEPTH is greater than the DEPTH
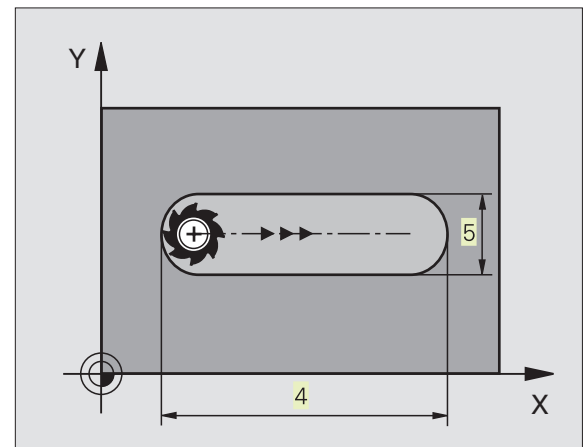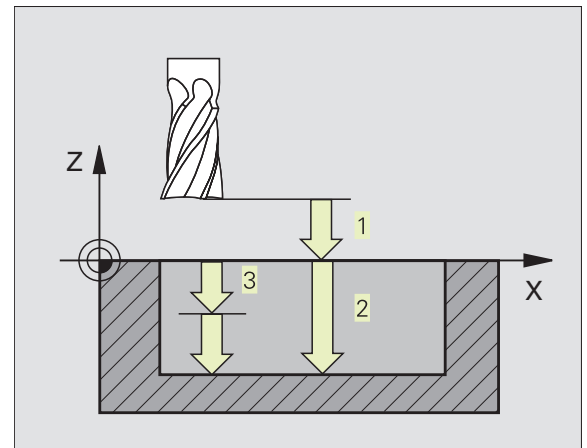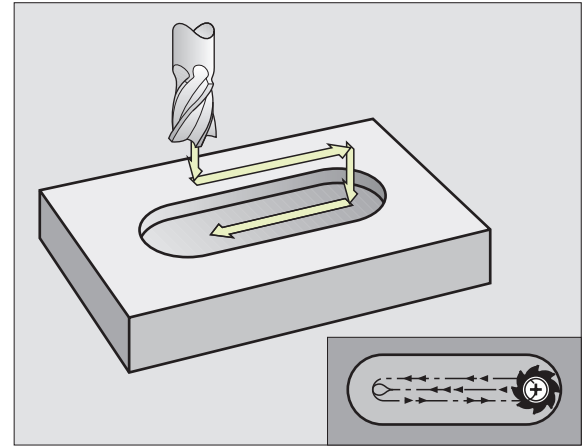
▶ FEED RATE FOR PECKING: Traversing speed of the tool during penetration

▶ FIRST SIDE LENGTH 4: Slot length; specify the sign to determine the first milling direction

▶ SECOND SIDE LENGTH 5: Slot width

▶ FEED RATE F: Traversing speed of the tool in the working plane

SLOT with reciprocating plunge-cut (Cycle 210)

☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

The cutter diameter must not be larger than the SLOT WIDTH and not smaller than a third of the SLOT WIDTH.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.

Roughing process

1 The TNC positions the tool in the tool axis to the 2ND SET-UP CLEARANCE and subsequently to the center of the left circle. From there, the TNC positions the tool to SET-UP CLEARANCE above the workpiece surface.

2 The tool moves at the ROUGHING FEED RATE to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot — plunge-cutting obliquely into the material — until it reaches the center of the right circle.

3 The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed MILLING DEPTH is reached.

4 At the MILLING DEPTH, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

Finishing process

5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).

6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE and — if programmed — to the 2ND SET-UP CLEARANCE.

▷ SET-UP CLEARANCE Q200 (incremental value):
Distance between tool tip and workpiece surface

▷ DEPTH Q201 (incremental value): Distance between
workpiece surface and bottom of slot

▷ FEED RATE FOR MILLING Q207: Traversing speed of
the tool during milling in mm/min.

▷ PECKING DEPTH Q202 (incremental value): Total
extent by which the tool is fed in the tool axis during a
reciprocating movement.

▷ MACHINING OPERATION (0/1/2) Q215:
Define the extent of machining:
0: Roughing and finishing
1: Roughing only
2: Finishing only

▷ WORKPIECE SURFACE COORDINATE Q203
(absolute value): Coordinate of the workpiece surface

▷ 2ND SET-UP CLEARANCE Q204 (incremental value): Z
coordinate at which no collision between tool and
workpiece (clamping devices) can occur.

▷ CENTER IN 1ST AXIS Q216 (absolute value): Center
of the slot in the main axis of the working plane

▷ CENTER IN 2ND AXIS Q217 (absolute value): Center
of the slot in the secondary axis of the working plane

▷ FIRST SIDE LENGTH Q218 (value parallel to the main
axis of the working plane): Enter the length of the slot

▷ SECOND SIDE LENGTH Q219 (value parallel to the
secondary axis of the working plane): Enter the slot
width. If you enter a slot width that equals the tool
diameter, the TNC will carry out the roughing process
only (slot milling).

▷ ANGLE OF ROTATION Q224 (absolute value): Angle
by which the entire slot is rotated. The center of
rotation lies in the center of the slot.
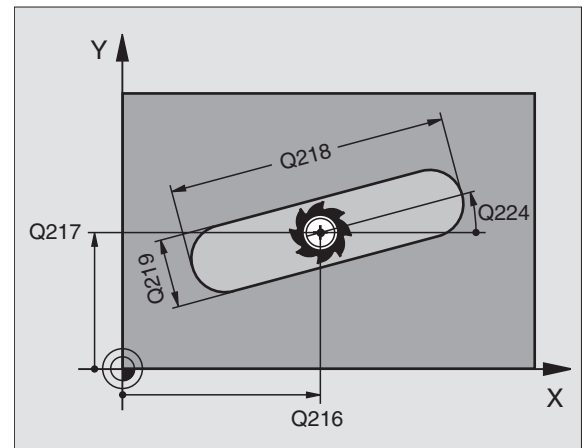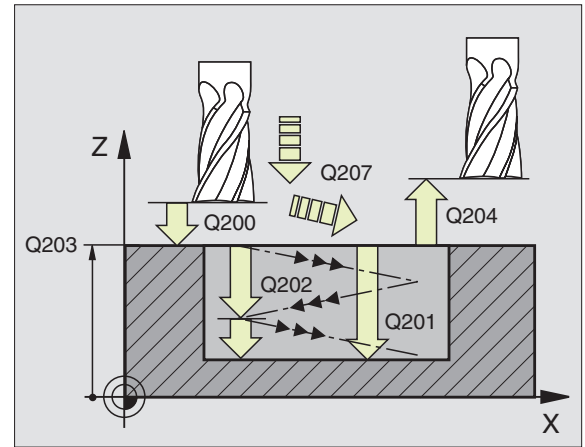
CIRCULAR SLOT with reciprocating plunge-cut (Cycle 211)

Roughing process

1 The TNC positions the tool in the tool axis to the 2ND SET-UP CLEARANCE and subsequently to the starting point. The control calculates the starting point from the entered cycle parameters. From there, the tool is positioned to the programmed SET-UP CLEARANCE above the workpiece surface.

2 The tool moves at the MILLING FEED RATE to the workpiece surface. From there, the cutter advances — plunge-cutting obliquely into the material — to the other end of the slot.

3 The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (2 to 3) is repeated until the programmed MILLING DEPTH is reached.

4 At the MILLING DEPTH, the TNC moves the tool for the purpose of face milling to the other end of the slot.
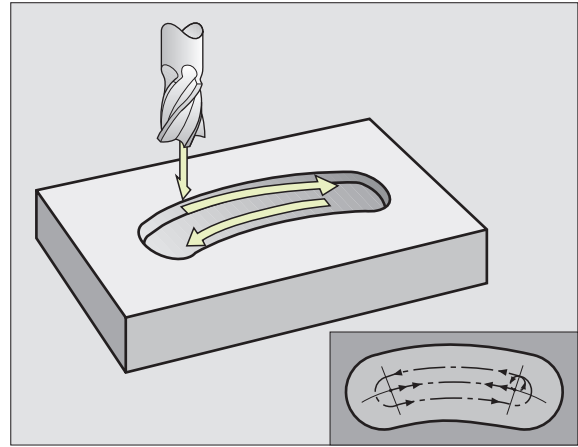
Finishing process

5 For finishing the slot, the TNC advances the tool tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).

6 When the tool reaches the end of the contour, it departs the contour tangentially.

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE and — if programmed — to the 2ND SET-UP CLEARANCE.
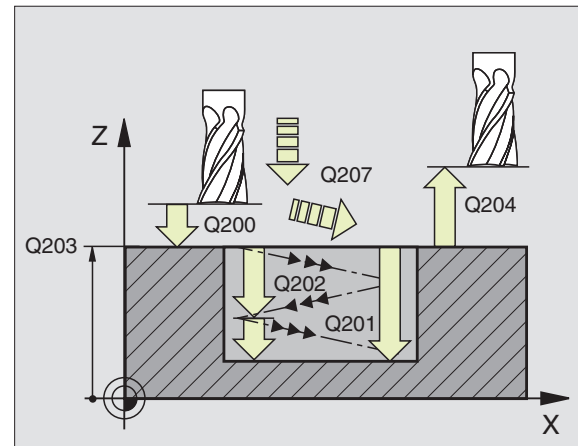
☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

The cutter diameter must not be larger than the SLOT WIDTH and not smaller than a third of the SLOT WIDTH.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface

▶ DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of slot

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool during milling in mm/min.

▶ PECKING DEPTH Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.

▶ MACHINING OPERATION (0/1/2) Q215:
Define the extent of machining:
0: Roughing and finishing
1: Roughing only
2: Finishing only

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the slot in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the slot in the secondary axis of the working plane

▶ PITCH CIRCLE DIAMETER Q244: Enter the diameter of the pitch circle

▶ SECOND SIDE LENGTH Q219: Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).

▶ STARTING ANGLE Q245 (absolute value): Enter the polar angle of the starting point.

▶ ANGULAR LENGTH Q248 (incremental value): Enter the angular length of the slot
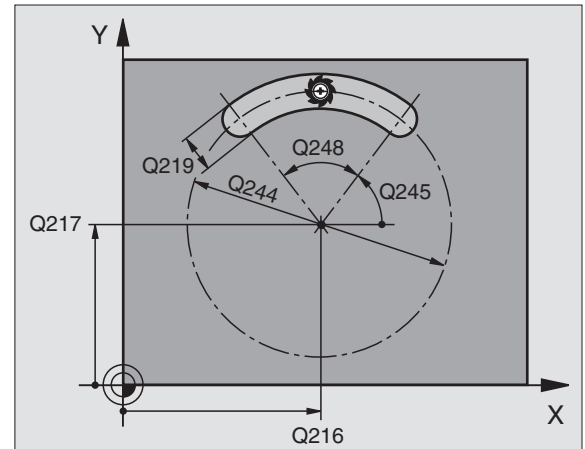
Example: Milling pockets, studs and slots
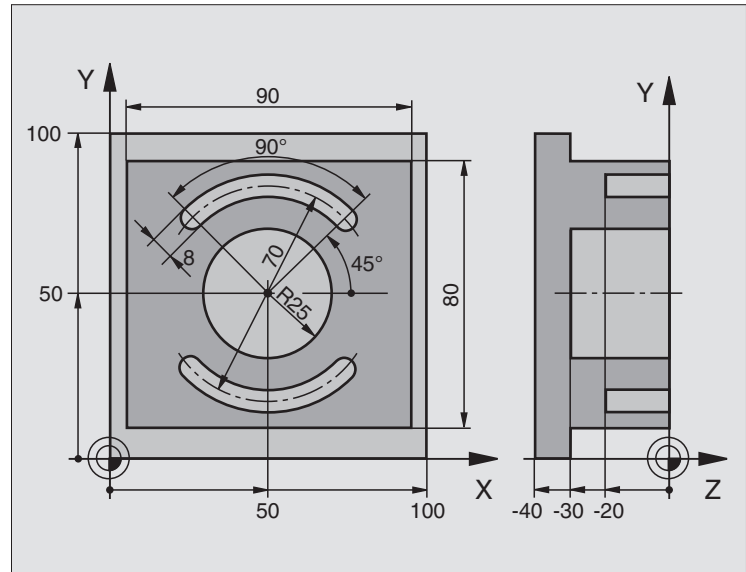
0	BEGIN PGM C210 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+6	Define tool for roughing/finishing
4	TOOL DEF 2 L+0 R+3	Define slotting mill
5	TOOL CALL 1 Z S3500	Call tool for roughing/finishing
6	L Z+250 R0 F MAX	Retract tool
7	CYCL DEF 213 STUD FINISHING	Define cycle for machining the contour outside
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-30 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q202=5 ;PECKING DEPTH	
	Q207=250 ;FEED RATE FOR MILLNG	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=20 ;2ND SET-UP CLEARANCE	
	Q216=+50 ;CENTER IN 1ST AXIS	
	Q217=+50 ;CENTER IN 2ND AXIS	
	Q218=90 ;FIRST SIDE LENGTH	
	Q219=80 ;SECOND SIDE LENGTH	
	Q220=0 ;CORNER RADIUS	
	Q221=5 ;ALLOWANCE IN 1ST AXS	
8	CYCL CALL M3	Call cycle for machining the contour outside

9	CYCL DEF 5.0 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
10	CYCL DEF 5.1 SET UP 2	
11	CYCL DEF 5.2 DEPTH -30	
12	CYCL DEF 5.3 PECKG 5 F250	
13	CYCL DEF 5.4 RADIUS 25	
14	CYCL DEF 5.5 F400 DR+	
15	L Z+2 R0 F MAX M99	Call CIRCULAR POCKET MILLING cycle
16	L Z+250 R0 F MAX M6	Tool change
17	TOOL CALL 2 Z S5000	Call slotting mill
18	CYCL DEF 211 CIRCULAR SLOT	Define cycle for slot 1
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-20 ;DEPTH	
	Q207=250 ;FEED RATE FOR MILLNG	
	Q202=5 ;PECKING DEPTH	
	Q215=0 ;MACHINING OPERATION	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
	Q216=+50 ;CENTER IN 1ST AXIS	
	Q217=+50 ;CENTER IN 2ND AXIS	
	Q244=70 ;PITCH CIRCLE DIA.	
	Q219=8 ;SECOND SIDE LENGTH	
	Q245=+45 ;STARTING ANGLE	
	Q248=90 ;ANGULAR LENGTH	
19	CYCL CALL M3	Call cycle for slot 1
20	FN 0: Q245 = +225	New starting angle for slot 2
21	CYCL CALL	Call cycle for slot 2
22	L Z+250 R0 F MAX M2	Retract tool, end of program
23	END PGM C210 MM	

8.4 Cycles for Machining Hole Patterns

The TNC provides two cycles for machining hole patterns:

Cycle	Soft key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

Cycle 1	PECKING
Cycle 2	TAPPING with a floating tap holder
Cycle 3	SLOT MILLING
Cycle 4	POCKET MILLING
Cycle 5	CIRCULAR POCKET MILLING
Cycle 17	RIGID TAPPING
Cycle 18	THREAD CUTTING

Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL MILLING CYCLE
Cycle 212	POCKET FINISHING
Cycle 213	STUD FINISHING
Cycle 214	CIRCULAR POCKET FINISHING
Cycle 215	CIRCULAR STUD FINISHING

CIRCULAR PATTERN (Cycle 220)

1 The TNC moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:
■ Move to 2ND SET-UP CLEARANCE (tool axis)
■ Approach starting point in the working plane
■ Move to SET-UP CLEARANCE above the workpiece surface (tool axis)

2 From this position, the TNC executes the last defined fixed cycle.

3 The tool then approaches the starting point for the next machining operation on a straight line at SET-UP CLEARANCE (or 2ND SET-UP CLEARANCE).

4 This process (1 to 3) is repeated until all machining operations have been executed.

☞ **Before programming, note the following:**

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

If you combine Cycle 220 with one of the fixed cycles 200 to 215, the SET-UP CLEARANCE, workpiece surface and 2ND SET-UP CLEARANCE that you defined in Cycle 220 will be effective for the selected fixed cycle.

▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the pitch circle in the main axis of the working plane

▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the pitch circle in the secondary axis of the working plane

▶ PITCH CIRCLE DIAMETER Q244: Diameter of the pitch circle

▶ STARTING ANGLE Q245 (absolute value): Angle between the main axis of the working plane and the starting point for the first machining operation on the pitch circle

▶ STOPPING ANGLE Q246 (absolute value): Angle between the main axis of the working plane and the starting point for the last machining operation on the pitch circle. Do not enter the same value for the STOPPING ANGLE and STARTING ANGLE. If you enter the STOPPING ANGLE greater than the START-ING ANGLE, machining will be carried out counter-clockwise; otherwise, machining will be clockwise.

▶ STEPPING ANGLE Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter a STEPPING ANGLE of 0, the TNC will cal-culate the STEPPING ANGLE from the STARTING and STOPPING ANGLES. If you enter a value other than 0, the TNC will not take the STOPPING ANGLE into account. The sign for the STEPPING ANGLE determines the working direction (– = clockwise).
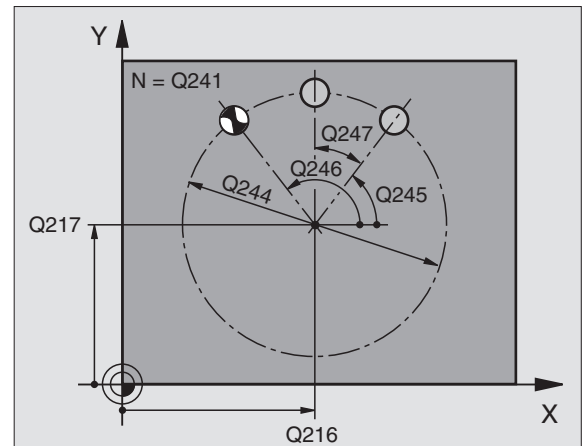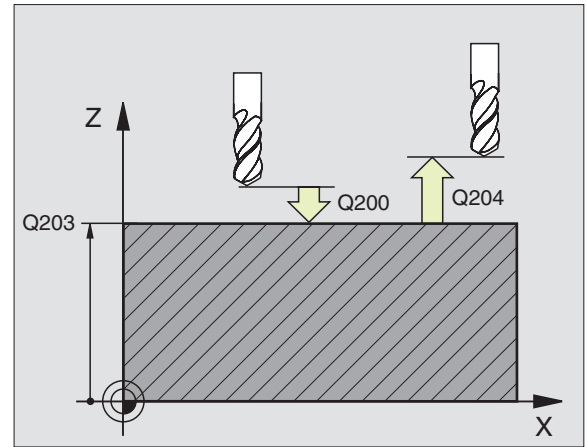
▷ NUMBER OF REPETITIONS Q241: Number of machining operations on a pitch circle

▷ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.

▷ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▷ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

LINEAR PATTERN (Cycle 221)

☞ **Before programming, note the following:**

Cycle 221 is DEF active, which means that Cycle 221 calls the last defined fixed cycle.

If you combine Cycle 221 with one of the fixed cycles 200 to 215, the SET-UP CLEARANCE, workpiece surface and 2ND SET-UP CLEARANCE that you defined in Cycle 221 will be effective for the selected fixed cycle.

1 The TNC automatically moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:
■ Move to 2ND SET-UP CLEARANCE (tool axis)
■ Approach starting point in the working plane
■ Move to SET-UP CLEARANCE above the workpiece surface (tool axis)

2 From this position, the TNC executes the last defined fixed cycle.

3 The tool then approaches the starting point for the next machining operation in the positive main axis direction at SET-UP CLEARANCE (or 2ND SET-UP CLEARANCE).

4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.

5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

6 From this position, the tool approaches the starting point for the next machining operation in the negative main axis direction.

7 This process (5 to 6) is repeated until all machining operations in the second line have been executed.
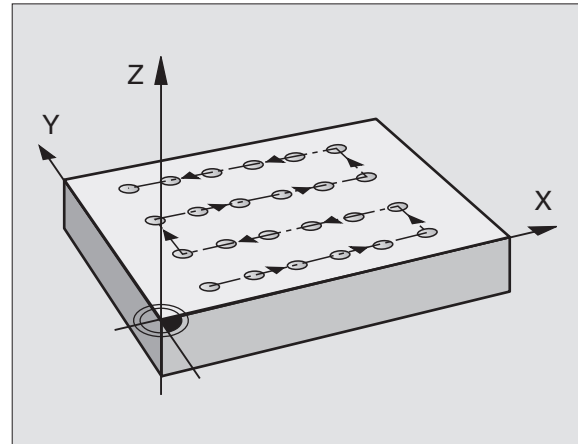
8 The tool then moves to the starting point of the next line.

9 All subsequent lines are processed in a reciprocating movement.

▶ STARTING POINT 1ST AXIS Q225 (absolute value): Coordinate of the starting point in the main axis of the working plane

▶ STARTING POINT 2ND AXIS Q226 (absolute value): Coordinate of the starting point in the secondary axis of the working plane

▶ SPACING IN 1ST AXIS Q237 (incremental value): Spacing between the individual points on a line

▶ SPACING IN 2ND AXIS Q238 (incremental): Spacing between the individual lines

▶ NUMBER OF COLUMNS Q242: Number of machining operations on a line

▶ NUMBER OF LINES Q243: Number of passes

▶ ANGLE OF ROTATION Q224 (absolute value): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute value): Coordinate of the workpiece surface

▶ 2ND SET-UP CLEARANCE Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
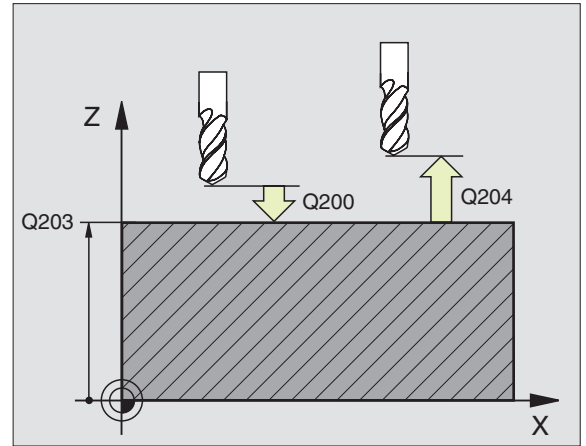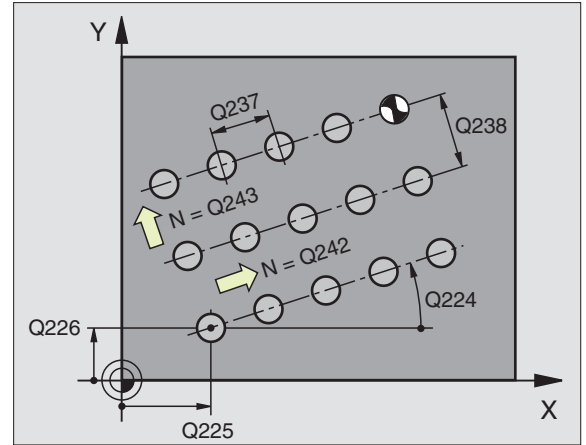
Example: Circular hole patterns
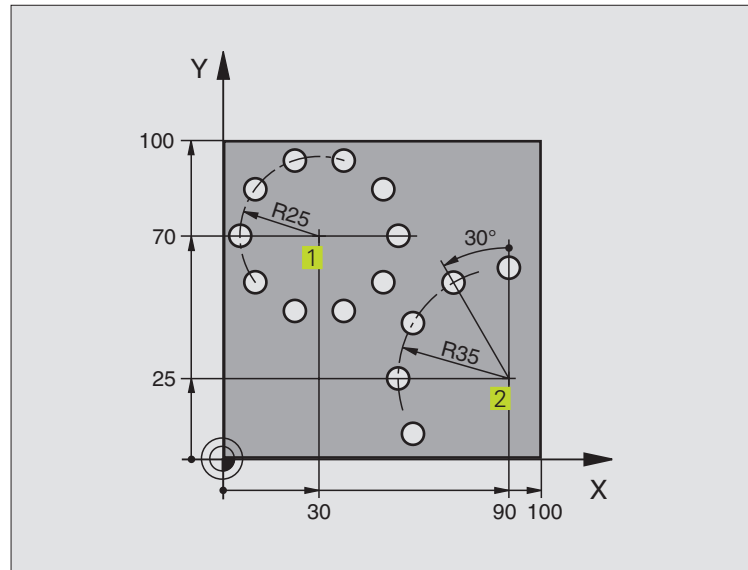
0	BEGIN PGM BOHRB MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+3	Define tool
4	TOOL CALL 1 Z S500	Call tool
5	L Z+250 R0 F MAX M3	Retract tool
6	CYCL DEF 200 DRILLING	Define DRILLING cycle
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q202=4 ;PECKING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	

7	CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 1, CYCL 200 is called automati-
	Q216=+30 ;CENTER IN 1ST AXIS	cally, Q200, Q203 and Q204 are effective as defined in Cycle 220
	Q217=+70 ;CENTER IN 2ND AXIS	
	Q244=50 ;PITCH CIRCLE DIA.	
	Q245=+0 ;STARTING ANGLE	
	Q246=+360 ;STOPPING ANGLE	
	Q247=+0 ;STEPPING ANGLE	
	Q241=10 ;NR OF REPETITIONS	
	Q200=2 ;SET-UP CLEARANCE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
8	CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automati-
	Q216=+90 ;CENTER IN 1ST AXIS	cally, Q200, Q203 and Q204 are effective as defined in Cycle 220
	Q217=+25 ;CENTER IN 2ND AXIS	
	Q244=70 ;PITCH CIRCLE DIA.	
	Q245=+90 ;STARTING ANGLE	
	Q246=+360 ;STOPPING ANGLE	
	Q247=+30 ;STEPPING ANGLE	
	Q241=5 ;NR OF REPETITIONS	
	Q200=2 ;SET-UP CLEARANCE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
9	L Z+250 R0 F MAX M2	Retract tool, end of program
10	END PGM BOHRB MM	

8.5 SL Cycles

SL cycles allow the contour-oriented machining of complex contours and achieve a particularly high degree of surface finish.

Characteristics of the contour
- A contour can be composed of several overlapping subcontours (up to 12 subcontours are possible). Islands and pockets can form a subcontour.

- The subcontour list (subprogram numbers) is entered in Cycle 14 CONTOUR GEOMETRY. The TNC calculates the contour from the subcontours.

- The individual subcontours are defined in subprograms.

- The memory capacity for programming an SL cycle is limited. All subprograms together can contain, for example, up to 128 straight-line blocks.

Characteristics of the subprograms
- Coordinate transformations are allowed.

- The TNC ignores feed rates F and miscellaneous functions M.

- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR.

- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.

- The subprograms must not contain tool axis coordinates.

- The working plane is defined in the first coordinate block of the subprogram. Parallel axes are allowed.

Characteristics of the fixed cycles
- The TNC automatically positions the tool to SET-UP CLEARANCE before a cycle.

- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.

- The radius of "inside corners" can be programmed — the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the ROUGH-OUT and SIDE-FINISHING cycles).

- The contour is approached in a tangential arc for side finishing.

- For floor finishing, the tool again approaches the workpiece in a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).

- The contour is machined throughout in either climb or up-cut milling.

☞ With MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle 20.

Overview of SL cycles

Cycle	Soft key
14 CONTOUR GEOMETRY (essential)	14 LBL 1...N
20 CONTOUR DATA (essential)	20 CONTOUR DATA
21 PILOT DRILLING (optional)	21
22 ROUGH-OUT (essential)	22
23 FLOOR FINISHING (optional)	23
24 SIDE FINISHING (optional)	24

Enhanced cycles:

Cycle	Soft key
25 CONTOUR TRAIN	25
27 CYLINDER SURFACE	27

Program structure: Working with SL cycles

```
0   BEGIN PGM SL2 MM
...
12  CYCL DEF 14.0 CONTOUR GEOMETRY ...
13  CYCL DEF 20.0 CONTOUR DATA ...
...
16  CYCL DEF 21.0 PILOT DRILLING ...
17  CYCL CALL
...
18  CYCL DEF 22.0 ROUGH-OUT ...
19  CYCL CALL
...
22  CYCL DEF 23.0 FLOOR FINISHING ...
23  CYCL CALL
...
26  CYCL DEF 24.0 SIDE FINISHING ...
27  CYCL CALL
...
50  L Z+250 R0 FMAX M2
51  LBL 1
...
60  LBL 0
61  LBL 2
...
62  LBL 0
...
99  END PGM SL2 MM
```

CONTOUR GEOMETRY (Cycle 14)

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.

☞ **Before programming, note the following:**

Cycle 14 is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subprograms (subcontours) in Cycle 14.


```
14
LBL 1...N
```

▶ LABEL NUMBERS FOR THE CONTOUR: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.
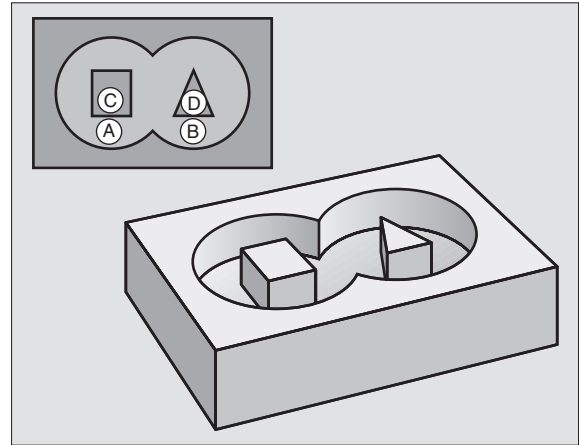
Overlapping contours

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: Overlapping pockets

☞ The subsequent programming examples are contour subprograms that are called by Cycle 14 CONTOUR GEOMETRY in a main program.

Pockets *A* and *B* overlap.

The TNC calculates the points of intersection S_1 and S_2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Left pocket

15	LBL 1
16	L X+10 Y+50 RR
17	CC X+35 Y+50
18	C X+10 Y+50 DR-
19	LBL 0

Subprogram 2: Right pocket

20	LBL 2
21	L X+90 Y+50 RR
22	CC X+65 Y+50
23	C X+90 Y+50 DR-
24	LBL 0
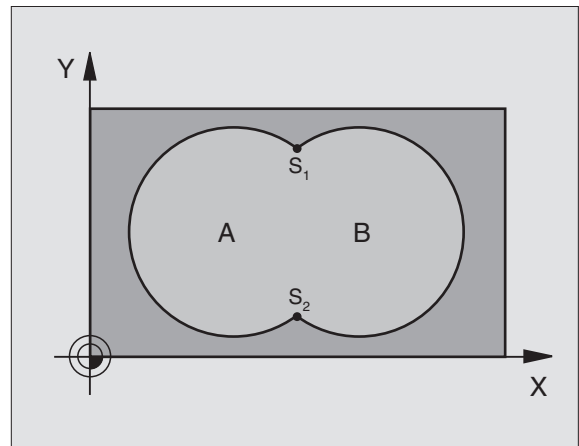
Area of inclusion

Both surfaces *A* and *B* are to be machined, including the mutually overlapped area:

■ The surfaces *A* and *B* must be pockets.

■ The first pocket (in Cycle 14) must start outside the second pocket.

Surface *A*:

15	LBL 1
16	L X+10 Y+50 RR
17	CC X+35 Y+50
18	C X+10 Y+50 DR-
19	LBL 0

Surface *B*:

20	LBL 2
21	L X+90 Y+50 RR
22	CC X+65 Y+50
23	C X+90 Y+50 DR-
24	LBL 0

Area of exclusion

Surface *A* is to be machined without the portion overlapped by *B*:

■ Surface *A* must be a pocket and *B* an island.

■ *A* must start outside of *B*.

Surface *A*:

15	LBL 1
16	L X+10 Y+50 RR
17	CC X+35 Y+50
18	C X+10 Y+50 DR-
19	LBL 0

Surface *B*:

20	LBL 2
21	L X+90 Y+50 RL
22	CC X+65 Y+50
23	C X+90 Y+50 DR-
24	LBL 0
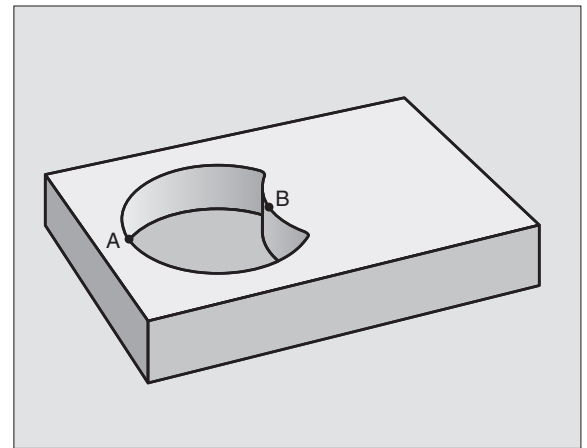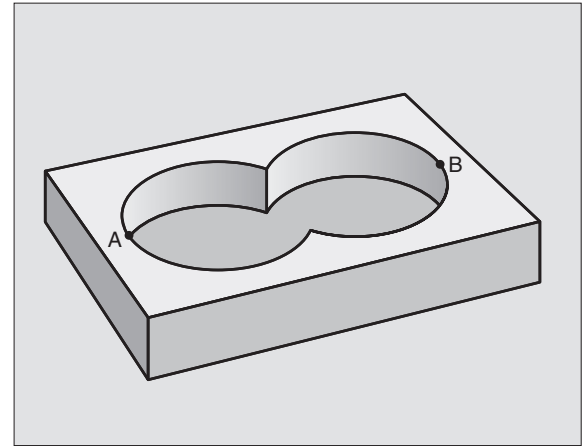
Area of intersection

Only the area overlapped by both *A* and *B* is to be machined. (The areas overlapped by *A* or *B* alone are to be left unmachined.)

■ *A* and *B* must be pockets.

■ *A* must start inside of *B*.

Surface *A*:

```
15   LBL 1
16   L X+60 Y+50 RR
17   CC X+35 Y+50
18   C X+60 Y+50 DR-
19   LBL 0
```

Surface *B*:

```
20   LBL 2
21   L X+90 Y+50 RR
22   CC X+65 Y+50
23   C X+90 Y+50 DR-
24   LBL 0
```
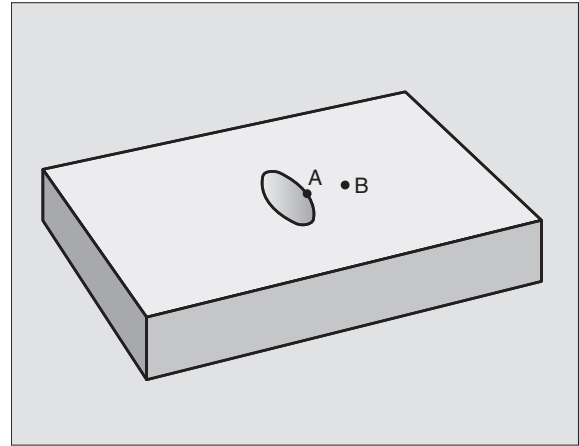
CONTOUR DATA (Cycle 20)

Machining data for the subprograms describing the subcontours are entered in Cycle 20.

☞ **Before programming, note the following:**

Cycle 20 is DEF active which means that it becomes effective as soon as it is defined in the part program.

The algebraic sign for the parameter DEPTH determines the working direction.

The machining data entered in Cycle 20 are valid for Cycles 21 to 24.

If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q19 cannot be used as program parameters.

▶ MILLING DEPTH Q1 (incremental value): Distance between workpiece surface and pocket floor

▶ PATH OVERLAP FACTOR Q2: Q2 x tool radius = stepover factor k

▶ FINISHING ALLOWANCE FOR SIDE Q3 (incremental value): Finishing allowance in the working plane

▶ FINISHING ALLOWANCE FOR FLOOR Q4 (incremental value): Finishing allowance in the tool axis

▶ WORKPIECE SURFACE COORDINATE Q5 (absolute value): Absolute coordinate of the workpiece surface
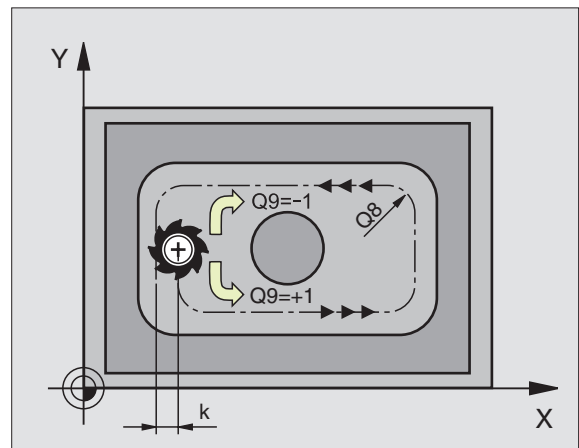
▶ SET-UP CLEARANCE Q6 (incremental value):
Distance between tool tip and workpiece surface

▶ CLEARANCE HEIGHT Q7 (absolute value): Absolute
height at which the tool cannot collide with the work-
piece (for intermediate positioning and retraction at the
end of the cycle)

▶ INSIDE CORNER RADIUS Q8: Inside "corner" round-
ing radius

▶ DIRECTION OF ROTATION Q9: Direction of rotation
for pockets:
■ Clockwise (Q9 = -1 up-cut milling for pocket and
island)
■ Counterclockwise (Q9 = +1 climb milling for pocket
and island)

You can check the machining parameters during a program inter-
ruption and overwrite them if required.
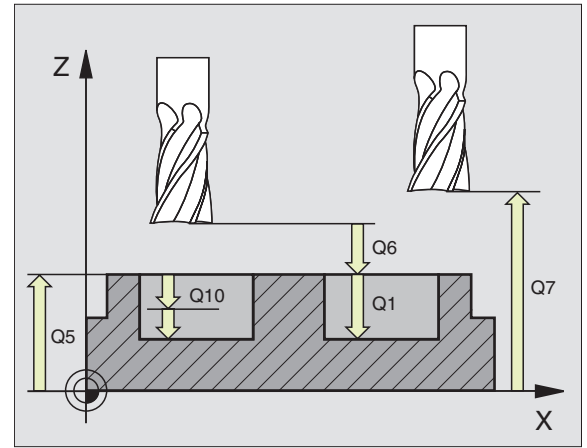
PILOT DRILLING (Cycle 21)

Process
Same as Cycle 1 Pecking (see page 133).

Application
Cycle 21 is for PILOT DRILLING of the cutter infeed points. It
accounts for the ALLOWANCE FOR SIDE and the ALLOWANCE
FOR FLOOR as well as the radius of the rough-out tool. The cutter
infeed points also serve as starting points for roughing.

▶ PECKING DEPTH Q10 (incremental value):
Dimension by which the tool drills in each infeed
(negative sign for negative working direction)

▶ FEED RATE FOR PECKING Q11: Traversing speed of
the tool in mm/min during drilling

▶ ROUGH-OUT TOOL NUMBER Q13: Tool number of
the roughing mill
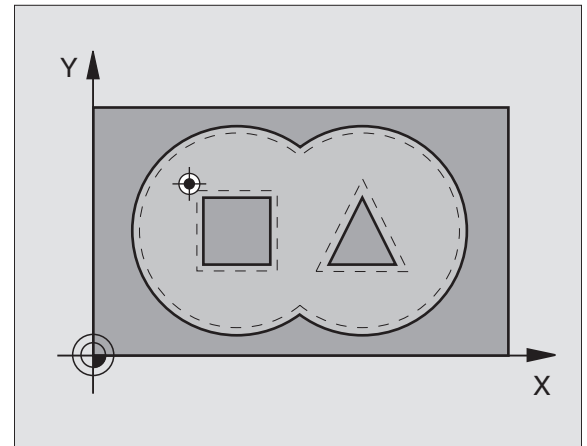
ROUGH-OUT (Cycle 22)

1 The TNC positions the tool over the cutter infeed point, taking the
ALLOWANCE FOR SIDE into account.

2 After reaching the first PECKING DEPTH, the tool mills the
contour in an outward direction at the programmed FEED RATE
FOR MILLING Q12.

3 First the island contours (*C* and *D* in the figure at right) are rough-
milled until the pocket contour (*A*, *B*) is approached.

4 Then the pocket contour is rough-milled and the tool is retracted
to the CLEARANCE HEIGHT.
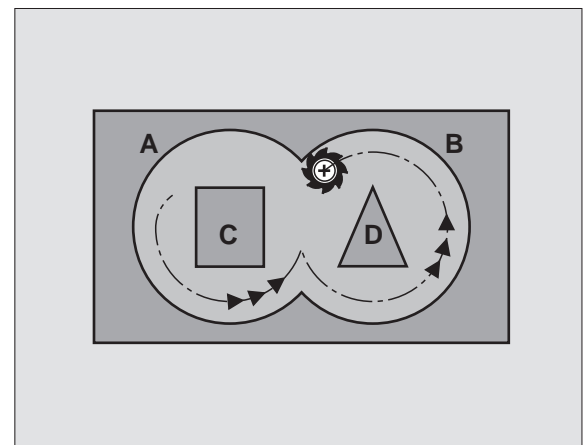
☞ **Before programming, note the following:**

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

▶ PECKING DEPTH Q10 (incremental value): Dimension by which the tool is plunged in each infeed

▶ FEED RATE FOR PECKING Q11: Traversing speed of the tool in mm/min during penetration

▶ FEED RATE FOR MILLING Q12: Traversing speed of the tool in mm/min while milling

▶ COARSE ROUGHING TOOL NUMBER Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If the contour has not been coarse-roughed, enter zero. If you enter a value other than 0, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool. If the portion that is to be fine-roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut. For this purpose, you must enter the tooth length LCUTS and the maximum plunge angle ANGLE of the tool in the tool table TOOL.T (see page 57). The TNC will otherwise generate an error message.

▶ RECIPROCATION FEED RATE Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut

FLOOR FINISHING (Cycle 23)

☞ The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The tool approaches the machining plane smoothly (in a vertically tangential arc). The tool then clears the finishing allowance remaining from rough-out.

▶ FEED RATE FOR PECKING Q11: Traversing speed of the tool during penetration

▶ FEED RATE FOR MILLING Q12: Traversing speed of the tool in the working plane
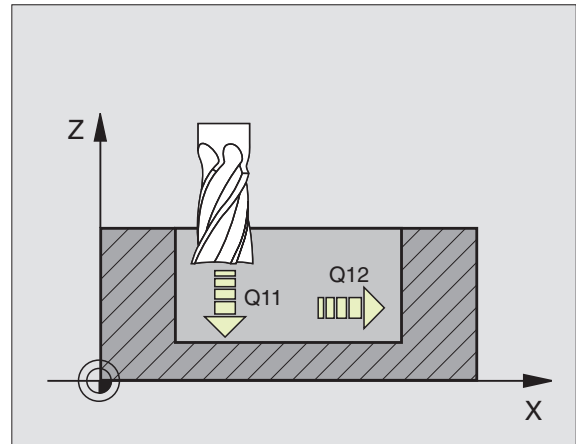
SIDE FINISHING (Cycle 24)

The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.

👉 **Before programming, note the following:**

The sum of ALLOWANCE FOR SIDE (Q14) and the radius of the finish mill must be smaller than the sum of ALLOWANCE FOR SIDE (Q3, Cycle 20) and the radius of the rough mill.

This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter "0" for the radius of the rough mill.

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

▶ DIRECTION OF ROTATION Q9: Direction of the cutter path: **+1**: Counterclockwise
　　　　　　　–1: Clockwise

▶ PECKING DEPTH Q10 (incremental value): Dimension by which the tool plunges in each infeed

▶ FEED RATE FOR PECKING Q11: Traversing speed of the tool during penetration

▶ FEED RATE FOR MILLING Q12: Traversing speed for milling

▶ FINISHING ALLOWANCE FOR SIDE Q14 (incremental value): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared.
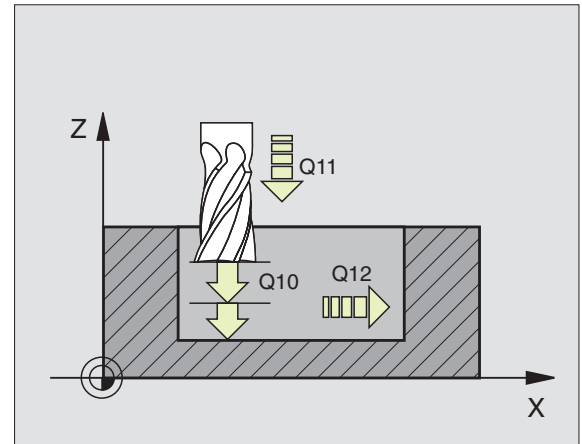
CONTOUR TRAIN (Cycle 25)

In conjunction with Cycle 14 CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (where the starting point of the contour is not the same as its end point).

Cycle 25 CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

■ The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.

■ If the radius of the selected tool is too large, the corners of the contour may have to be reworked.

■ The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.

■ The tool can traverse back and forth for milling in several infeeds: This results in faster machining.

■ Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.
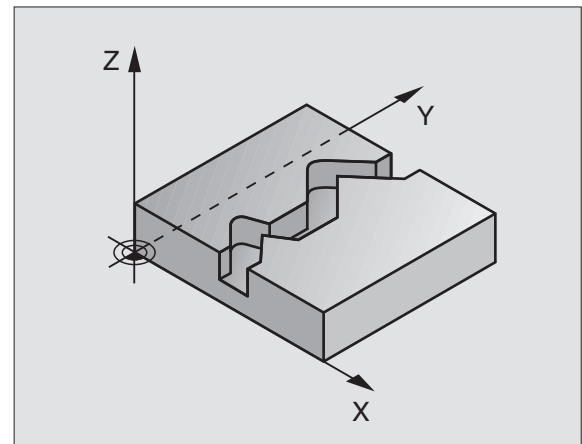
☞ **Before programming, note the following:**

The algebraic sign for the parameter DEPTH determines the working direction.

The TNC only takes the first label of Cycle 14 CONTOUR GEOMETRY into account.

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

Cycle 20 CONTOUR DATA is not required.

Positions that are programmed in incremental dimensions immediately after Cycle 25 are referenced to the position of the tool at the end of the cycle.

▶ MILLING DEPTH Q1 (incremental value): Distance between workpiece surface and contour floor

▶ FINISHING ALLOWANCE FOR SIDE Q3 (incremental value): Finishing allowance in the working plane

▶ WORKPIECE SURFACE COORDINATE Q5 (absolute value): Absolute coordinate of the workpiece surface referenced to the workpiece datum

▶ CLEARANCE HEIGHT Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle.

▶ PECKING DEPTH Q10 (incremental value): Dimension by which the tool is plunged for each infeed

▶ FEED RATE FOR PECKING Q11: Traversing speed of the tool in the tool axis

▶ FEED RATE FOR MILLING Q12: Traversing speed of the tool in the working plane

▶ CLIMB OR UP-CUT Q15:
Climb milling: Input value = +1
Conventional up-cut milling: Input value = –1
To enable climb milling and up-cut milling alternately in several infeeds: Input value = 0

CYLINDER SURFACE (Cycle 27)

The machine tool and the TNC must be prepared by the machine tool builder for the use of Cycle 27 CYLINDER SURFACE.

This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining.

The contour is described in a subprogram identified in Cycle 14 CONTOUR GEOMETRY.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, has the tool axis as its parallel axis. The path functions L, CHF, CR, RND are available for programming.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition.

☞ **Before programming, note the following:**

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

The algebraic sign for the parameter DEPTH determines the working direction.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle cannot be used in a tilted working plane.

▶ MILLING DEPTH Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour

▶ FINISHING ALLOWANCE FOR SIDE Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.

▶ SET-UP CLEARANCE Q6 (incremental value): Distance between the tool tip and the cylinder surface

▶ PECKING DEPTH Q10 (incremental value): Dimension by which the tool advances in each infeed

▶ FEED RATE FOR PECKING Q11: Traversing speed of the tool in the tool axis

▶ FEED RATE FOR MILLING Q12: Traversing speed of the tool in the working plane

▶ RADIUS Q16: Radius of the cylinder on which the contour is to be machined

▶ DIMENSION TYPE (ANG/LIN) Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)
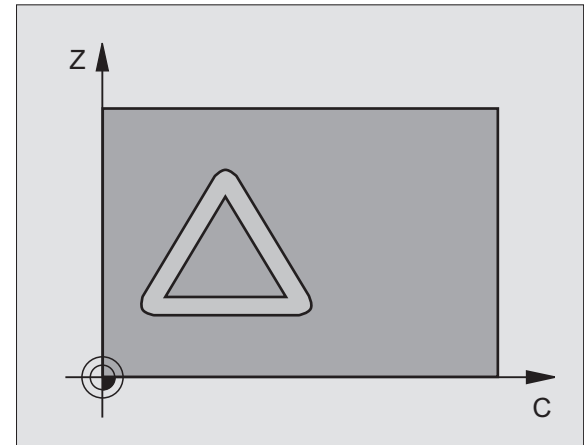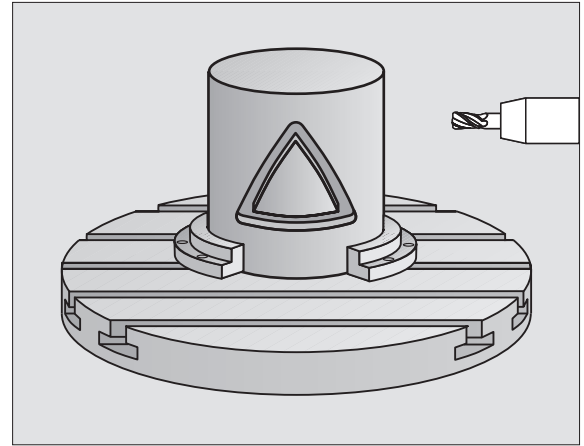
Example: Roughing-out and fine-roughing a pocket
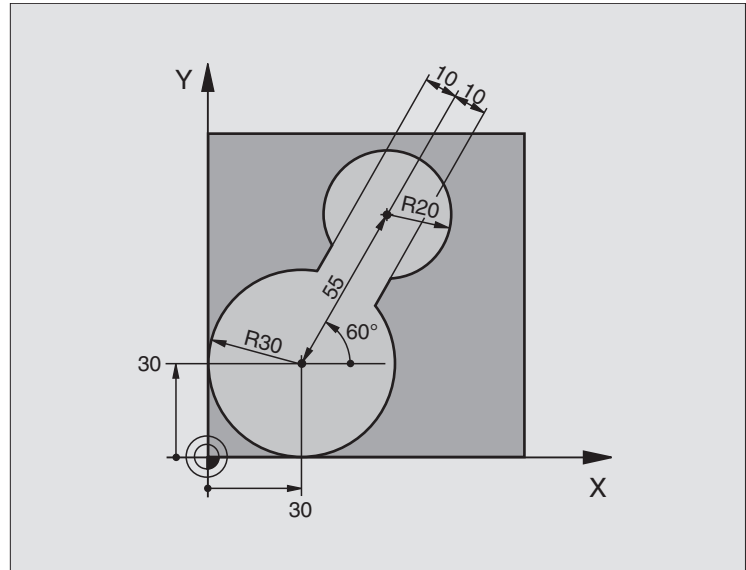
0	BEGIN PGM C20 MM	
1	BLK FORM 0.1 Z X-10 Y-10 Z-40	
2	BLK FORM 0.2 X+100 Y+100 Z+0	Define workpiece blank
3	TOOL DEF 1 L+0 R+15	Tool definition: coarse roughing tool
4	TOOL DEF 2 L+0 R+7.5	Tool definition: fine roughing tool
5	TOOL CALL 1 Z S2500	Tool call: coarse roughing tool
6	L Z+250 R0 F MAX	Retract tool
7	CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
8	CYCL DEF 14.1 CONTOUR LABEL 1	
9	CYCL DEF 20.0 CONTOUR DATA	Define general machining parameters
	Q1=-20 ;MILLING DEPTH	
	Q2=1 ;TOOL PATH OVERLAP	
	Q3=+0 ;ALLOWANCE FOR SIDE	
	Q4=+0 ;ALLOWANCE FOR FLOOR	
	Q5=+0 ;WORKPIECE SURFACE COORD.	
	Q6=2 ;SET-UP CLEARANCE	
	Q7=+100 ;CLEARANCE HEIGHT	
	Q8=0.1 ;ROUNDING RADIUS	
	Q9=-1 ;DIRECTION OF ROTATION	

10	CYCL DEF 22.0 ROUGH-OUT	Cycle definition: Coarse roughing
	Q10=5 ;PECKING DEPTH	
	Q11=100 ;FEED RATE FOR PECKING	
	Q12=350 ;FEED RATE FOR MILLING	
	Q18=0 ;COARSE ROUGHING TOOL	
	Q19=150 ;RECIPROCATION FEED RATE	
11	CYCL CALL M3	Cycle call: Coarse roughing
12	L Z+250 R0 F MAX M6	Tool change
13	TOOL CALL 2 Z S3000	Tool call: fine roughing tool
14	CYCL DEF 22.0 ROUGH-OUT	Cycle definition: Fine roughing
	Q10=5 ;PECKING DEPTH	
	Q11=100 ;FEED RATE FOR PECKING	
	Q12=350 ;FEED RATE FOR MILLING	
	Q18=1 ;COARSE ROUGHING TOOL	
	Q19=150 ;RECIPROCATION FEED RATE	
15	CYCL CALL M3	Cycle call: Fine roughing
16	L Z+250 R0 F MAX M2	Retract tool, end of program
17	LBL 1	Contour subprogram
18	L X+0 Y+30 RR	(see second FK programming example on page 111)
19	FC DR- R30 CCX+30 CCY+30	
20	FL AN+60 PDX+30 PDY+30 D10	
21	FSELECT 3	
22	FPOL X+30 Y+30	
23	FC DR- R20 CCPR+55 CCPA+60	
24	FSELECT 2	
25	FL AN-120 PDX+30 PDY+30 D10	
26	FSELECT 3	
27	FC X+0 DR- R30 CCX+30 CCY+30	
28	FSELECT 2	
29	LBL 0	
30	END PGM C20 MM	

Example: Pilot drilling, roughing-out and finishing overlapping contours
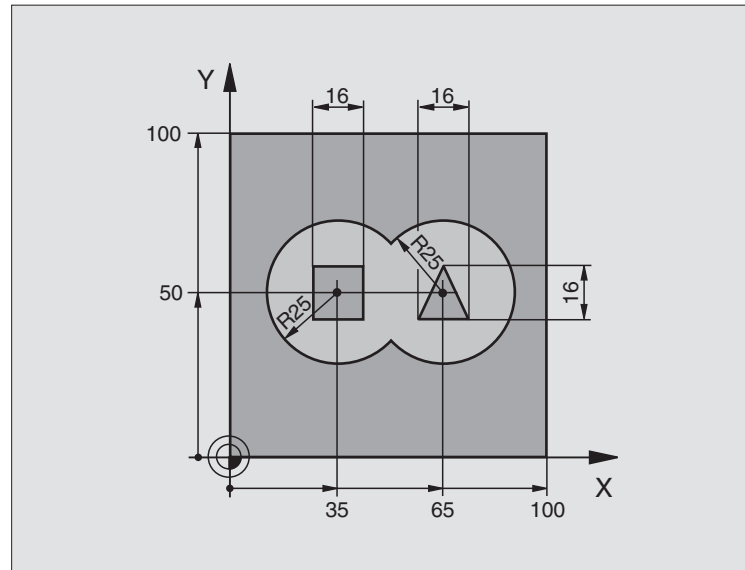
0	BEGIN PGM C21 MM	
1	BLK FORM 0.1 Z X-10 Y-10 Z-40	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+6	Tool definition: drill
4	TOOL DEF 2 L+0 R+6	Tool definition: rough mill / finish mill
5	TOOL CALL 1 Z S2500	Tool call: drill
6	L Z+250 R0 F MAX	Retract tool
7	CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
8	CYCL DEF 14.1 CONTOUR LABEL 1 /2 /3 /4	
9	CYCL DEF 20.0 CONTOUR DATA	Define general machining parameters
	Q1=-20 ;MILLING DEPTH	
	Q2=1 ;TOOL PATH OVERLAP	
	Q3=+0.5 ;ALLOWANCE FOR SIDE	
	Q4=+0.5 ;ALLOWANCE FOR FLOOR	
	Q5=+0 ;WORKPIECE SURFACE COORD.	
	Q6=2 ;SET-UP CLEARANCE	
	Q7=+100 ;CLEARANCE HEIGHT	
	Q8=0.1 ;ROUNDING RADIUS	
	Q9=-1 ;DIRECTION OF ROTATION	
10	CYCL DEF 21.0 PILOT DRILLING	Cycle definition: PILOT DRILLING
	Q10=5 ;PECKING DEPTH	
	Q11=250 ;FEED RATE FOR PECKING	
	Q13=2 ;ROUGH-OUT TOOL	
11	CYCL CALL M3	Cycle call: PILOT DRILLING

12	`L Z+250 R0 F MAX M6`	Tool change
13	`TOOL CALL 2 Z S3000`	Tool call: rough mill / finish mill
14	`CYCL DEF 22.0 ROUGH-OUT`	Cycle definition: ROUGH-OUT
	`Q10=5    ;PECKING DEPTH`	
	`Q11=100  ;FEED RATE FOR PECKING`	
	`Q12=350  ;FEED RATE FOR MILLING`	
	`Q18=0    ;COARSE ROUGHING TOOL`	
	`Q19=150  ;RECIPROCATION FEED RATE`	
15	`CYCL CALL M3`	Cycle call: ROUGH-OUT
16	`CYCL DEF 23.0 FLOOR FINISHING`	Cycle definition: FLOOR FINISHING
	`Q11=100  ;FEED RATE FOR PECKING`	
	`Q12=200  ;FEED RATE FOR MILLING`	
17	`CYCL CALL`	Cycle call: FLOOR FINISHING
18	`CYCL DEF 24.0 SIDE FINISHING`	Cycle definition: SIDE FINISHING
	`Q9=+1    ;DIRECTION OF ROTATION`	
	`Q10=5    ;PECKING DEPTH`	
	`Q11=100  ;FEED RATE FOR PECKING`	
	`Q12=400  ;FEED RATE FOR MILLING`	
	`Q14=+0   ;ALLOWANCE FOR SIDE`	
19	`CYCL CALL`	Cycle call: SIDE FINISHING
20	`L Z+250 R0 F MAX M2`	Retract tool, end of program
21	`LBL 1`	Contour subprogram 1: left pocket
22	`CC X+35 Y+50`	
23	`L X+10 Y+50 RR`	
24	`C X+10 DR-`	
25	`LBL 0`	
26	`LBL 2`	Contour subprogram 2: right pocket
27	`CC X+65 Y+50`	
28	`L X+90 Y+50 RR`	
29	`C X+90 DR-`	
30	`LBL 0`	
31	`LBL 3`	Contour subprogram 3: square left island
32	`L X+27 Y+50 RL`	
33	`L Y+58`	
34	`L X+43`	
35	`L Y+42`	
36	`L X+27`	
37	`LBL 0`	
38	`LBL 4`	Contour subprogram 4: triangular right island
39	`L X+65 Y+42 RL`	
40	`L X+57`	
41	`L X+65 Y+58`	
42	`L X+73 Y+42`	
43	`LBL 0`	
44	`END PGM C21 MM`	

Example: Contour train
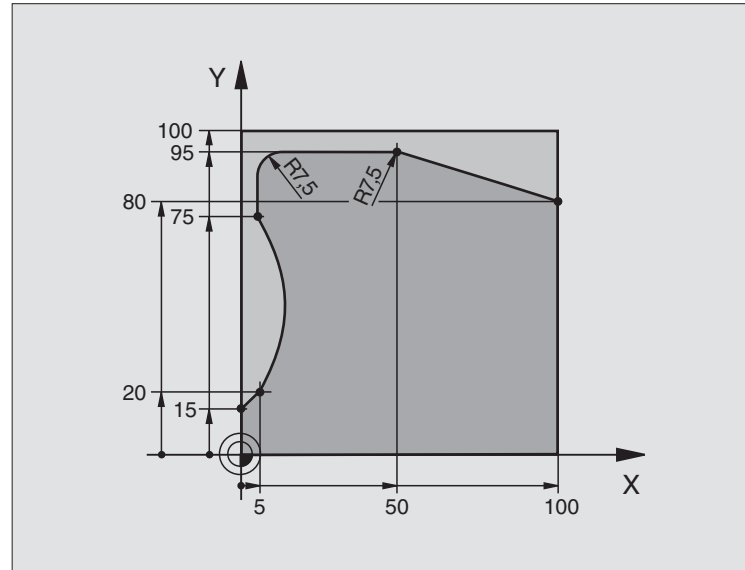
0	BEGIN PGM C25 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define workpiece blank
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define tool
4	TOOL CALL 1 Z S2000	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
7	CYCL DEF 14.1 CONTOUR LABEL 1	
8	CYCL DEF 25.0 CONTOUR TRAIN	Define machining parameters
	Q1=-20 ;MILLING DEPTH	
	Q3=+0 ;ALLOWANCE FOR SIDE	
	Q5=+0 ;WORKPIECE SURFACE COORD.	
	Q7=+250 ;CLEARANCE HEIGHT	
	Q10=5 ;PECKING DEPTH	
	Q11=100 ;FEED RATE FOR PECKING	
	Q12=200 ;FEED RATE FOR MILLING	
	Q15=+1 ;CLIMB OR UP-CUT	
9	CYCL CALL M3	Cycle call
10	L Z+250 R0 F MAX M2	Retract tool, end of program

11	LBL 1	Contour subprogram
12	L X+0 Y+15 RL	
13	L X+5 Y+20	
14	CT X+5 Y+75	
15	L Y+95	
16	RND R7.5	
17	L X+50	
18	RND R7.5	
19	L X+100 Y+80	
20	LBL 0	
21	END PGM C25 MM	

Example: Cylinder surface
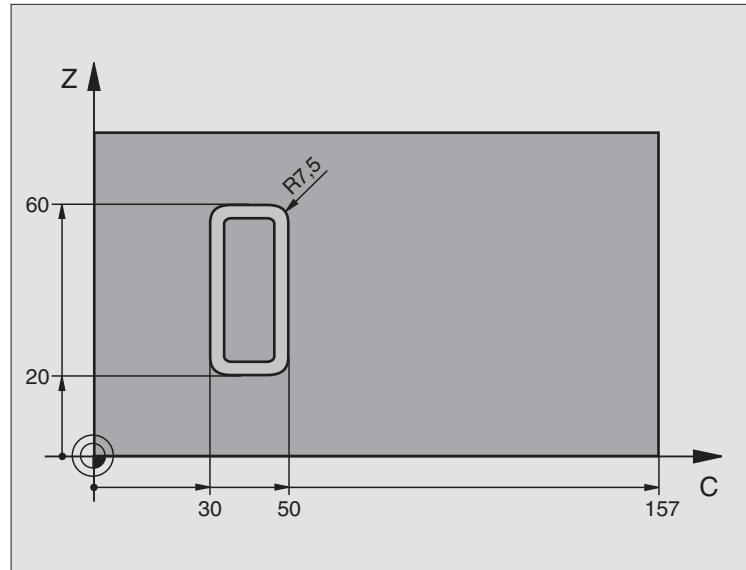
0	BEGIN PGM C27 MM	
1	TOOL DEF 1 L+0 R+3.5	Define tool
2	TOOL CALL 1 Y S2000	Call tool, tool axis is Y
3	L Y+250 R0 F MAX	Retract tool
4	CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
5	CYCL DEF 14.1 CONTOUR LABEL 1	
6	CYCL DEF 27.0 CYLINDER SURFACE	Define machining parameters
	Q1=-7 ;MILLING DEPTH	
	Q3=+0 ;ALLOWANCE FOR SIDE	
	Q6=2 ;SET-UP CLEARANCE	
	Q10=4 ;PECKING DEPTH	
	Q11=100 ;FEED RATE FOR PECKING	
	Q12=250 ;FEED RATE FOR MILLING	
	Q16=25 ;RADIUS	
	Q17=1 ;DIMENSION TYPE (ANG/LIN)	
7	L C+0 R0 F MAX M3	Pre-position rotary table
8	CYCL CALL	Cycle call
9	L Y+250 R0 F MAX M2	Retract tool, end of program

10	LBL 1	Contour subprogram
11	L C+40 Z+20 RL	Data for the rotary axis are entered in mm (Q17=1)
12	L C+50	
13	RND R7.5	
14	L Z+60	
15	RND R7.5	
16	L IC-20	
17	RND R7.5	
18	L Z+20	
19	RND R7.5	
20	L C+40	
21	LBL 0	
22	END PGM C27 MM	

8.6 Cycles for Multipass Milling

The TNC offers three cycles for machining the following surface types:

■ Surfaces generated by digitizing

■ Flat, rectangular surfaces

■ Flat, oblique-angled surfaces

■ Surfaces that are inclined in any way

■ Twisted surfaces

Cycle	Soft key
30 RUN DIGITIZED DATA For multipass milling of digitized surface data in several infeeds	30 MILL PNT-DAT
230 MULTIPASS MILLING For flat, rectangular surfaces	230
231 RULED SURFACE For oblique, inclined or twisted surfaces	231

RUN DIGITIZED DATA (Cycle 30)

1 From the current position, the TNC positions the tool in rapid traverse FMAX in the tool axis to the SET-UP CLEARANCE above the MAX point that you have programmed in the cycle.

2 The tool then moves in FMAX in the working plane to the MIN point you have programmed in the cycle.

3 From this point, the tool advances to the first contour point at the FEED RATE FOR PECKING.

4 The TNC subsequently processes all points that are stored in the digitizing data file at the FEED RATE FOR MILLING. If necessary, the TNC retracts the tool between machining operations to SET-UP CLEARANCE if specific areas are to be left unmachined.

5 At the end of the cycle, the tool is retracted in FMAX to SET-UP CLEARANCE.

Before programming, note the following:

You can use Cycle 30 to run digitizing data and PNT files.

If you want to run PNT files in which no tool axis coordinate is programmed, the milling depth is derived from the programmed MIN point in the tool axis.

`30 MILL PNT-DAT`

▶ PGM NAME DIGITIZING DATA: Enter the name of the file in which the digitizing data is stored. If the file is not stored in the current directory, enter the complete path.

▶ MIN. POINT RANGE: Lowest coordinates (X, Y and Z coordinates) in the range to be milled

▶ MAX. POINT RANGE: Highest coordinates (X, Y and Z coordinates) in the range to be milled

▶ SET-UP CLEARANCE 1 (incremental value): Distance between tool tip and workpiece surface for tool movements in rapid traverse

▶ PECKING DEPTH 2 (incremental value): Dimension by which the tool is advanced in each infeed

▶ FEED RATE FOR PECKING 3: Traversing speed of the tool in mm/min during penetration

▶ FEED RATE FOR MILLING 4: Traversing speed of the tool in mm/min while milling

▶ MISCELLANEOUS FUNCTION M: Optional entry of a miscellaneous function, for example M112
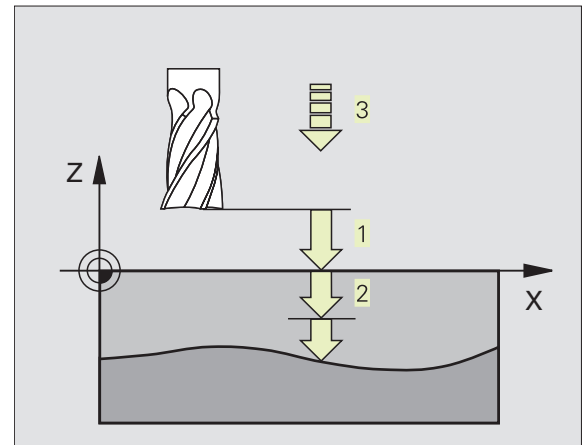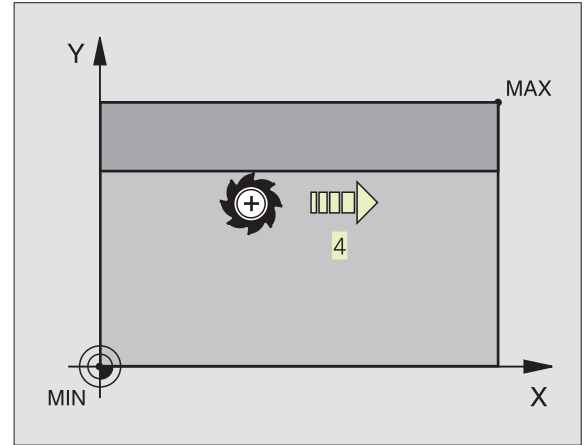
MULTIPASS MILLING (Cycle 230)

1 From the current position, the TNC positions the tool in rapid traverse FMAX in the working plane to the starting point 1. In the process, the TNC offsets the tool by the tool radius upwards and to the left.

2 The tool then moves in FMAX in the tool axis to SET-UP CLEARANCE. From there it approaches the programmed starting position in the tool axis at the FEED RATE FOR PLUNGING.

3 The tool subsequently advances to the stopping point 2 at the FEED RATE FOR MILLING. The stopping point is calculated from the programmed starting point, the programmed length and the tool radius.

4 The TNC offsets the tool to the starting point in the next pass at the STEPOVER FEED RATE. The offset is calculated from the programmed width and the number of cuts.

5 The tool then returns to the starting point 1 in the negative direction.

6 Multipass milling is repeated until the programmed surface has been completed.

7 At the end of the cycle, the tool is retracted in FMAX to SET-UP CLEARANCE.
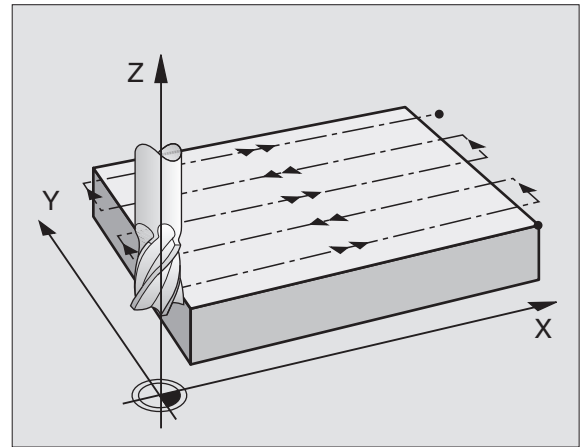
☞ **Before programming, note the following:**

From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

▶ STARTING POINT IN 1ST AXIS Q225 (absolute value): MIN point coordinate of the surface to be multipass-milled in the main axis of the working plane

▶ STARTING POINT IN 2ND AXIS Q226 (absolute value): MIN point coordinate of the surface to be multipass-milled in the secondary axis of the working plane

▶ STARTING POINT IN 3RD AXIS Q227 (absolute value): Height in the spindle axis at which multipass-milling is carried out.

▶ FIRST SIDE LENGTH Q218 (incremental value): Length of the surface to be multipass-milled in the main axis of the working plane, referenced to the STARTING POINT IN 1ST AXIS

▶ SECOND SIDE LENGTH Q219 (incremental value): Length of the surface to be multipass-milled in the secondary axis of the working plane, referenced to the STARTING POINT IN 2ND AXIS

▶ NUMBER OF CUTS Q240: Number of passes to be made over the width

▶ FEED RATE FOR PLUNGING Q206: Traversing speed of the tool in mm/min when moving from SET-UP CLEARANCE to the milling depth

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool in mm/min while milling.

▶ STEPOVER FEED RATE Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207 If you are moving it transversely in the open, Q209 may be greater than Q207.

▶ SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.
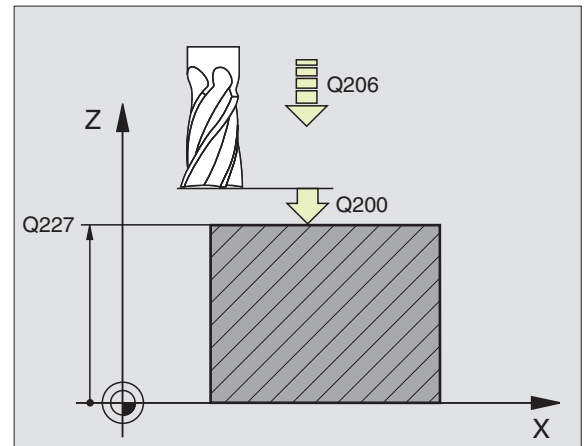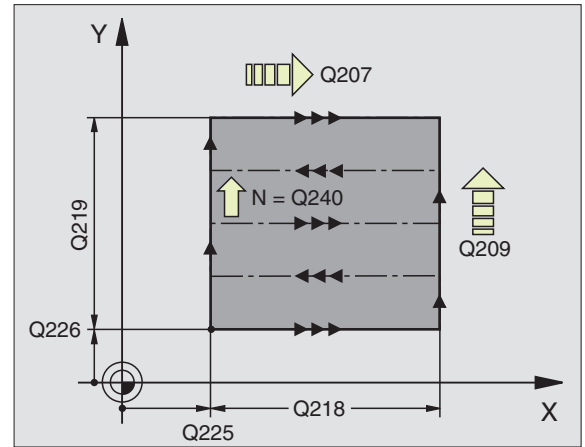
RULED SURFACE (Cycle 231)

1 From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1.

2 The tool then moves to the stopping point 2 at the FEED RATE FOR MILLING.

3 From this point, the tool moves in rapid traverse FMAX by the tool diameter in the positive tool axis direction, and then back to starting point 1.

4 At starting point 1, the tool again advances by the last used Z value.

5 The TNC subsequently offsets the tool in all three axes from point 1 in the direction of point 4 to the next pass.

6 From this point, the tool moves to the stopping point on this pass. The TNC calculates the stopping point from point 2 and an offset in the direction of point 3.

7 Multipass milling is repeated until the programmed surface has been completed.

8 At the end of the cycle, the tool is positioned above the highest programmed point in the tool axis, offset by the tool diameter.

Cutting motion
You can freely choose the starting point and thus the milling direction since the TNC always performs the individual cuts from point 1 to point 2 and the process sequence is executed from points 1/2 to points 3/4. You can position point 1 in any corner of the surface to be machined.

If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

■ a shaping cut (tool axis coordinate of point 1 greater than tool axis coordinate of point 2) for slightly inclined surfaces, or

■ a drawing cut (tool axis coordinate of point 1 less than tool axis coordinate of point 2) for steep surfaces

■ When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) parallel to the direction of the steepest inclination — see figure at center right.

If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way:

■ When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) perpendicular to the direction of the steepest inclination — see figure at lower right.
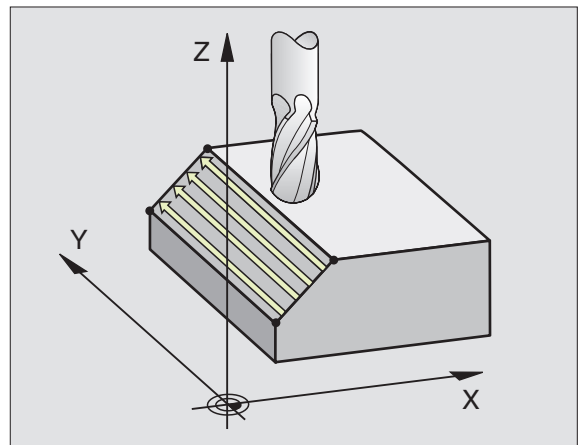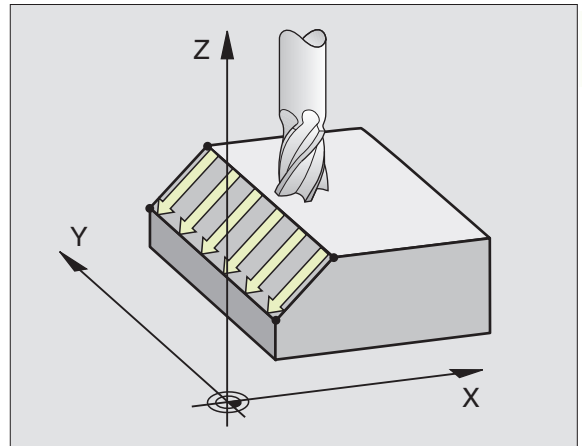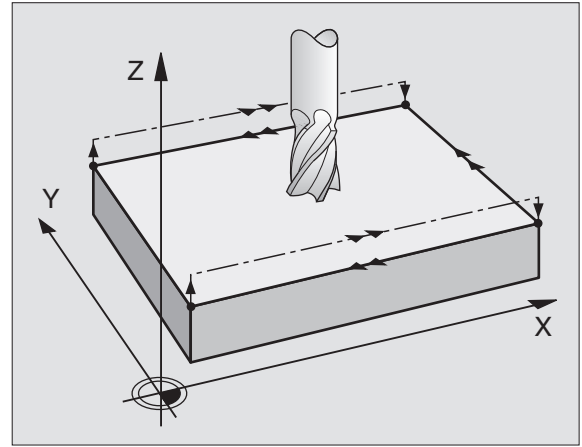
☞ **Before programming, note the following:**

From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

The TNC moves the tool with RADIUS COMPENSATION R0 to the programmed positions.

If required, use a center-cut end mill (ISO 1641).

▶ STARTING POINT IN 1ST AXIS Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the main axis of the working plane

▶ STARTING POINT IN 2ND AXIS Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the secondary axis of the working plane

▶ STARTING POINT IN 3RD AXIS Q227 (absolute value): Starting point coordinate of the surface to be multipass-milled in the tool axis

▶ 2ND POINT IN 1ST AXIS Q228 (absolute value): Stopping point coordinate of the surface to be multi-pass milled in the main axis of the working plane

▶ 2ND POINT IN 2ND AXIS Q229 (absolute value): Stopping point coordinate of the surface to be multi-pass milled in the secondary axis of the working plane

▶ 2ND POINT IN 3RD AXIS Q230 (absolute value): Stopping point coordinate of the surface to be multi-pass milled in the tool axis

▶ 3RD POINT IN 1ST AXIS Q231 (absolute): Coordinate of point 3 in the main axis of the working plane

▶ 3RD POINT IN 2ND AXIS Q232 (absolute): Coordinate of point 3 in the secondary axis of the working plane

▶ 3RD POINT IN 3RD AXIS Q233 (absolute): Coordinate of point 3 in the tool axis

▶ 4TH POINT IN 1ST AXIS Q234 (absolute): Coordinate of point 4 in the main axis of the working plane

▶ 4TH POINT IN 2ND AXIS Q235 (absolute): Coordinate of point 4 in the secondary axis of the working plane

▶ 4TH POINT IN 3RD AXIS Q236 (absolute): Coordinate of point 4 in the tool axis

▶ NUMBER OF CUTS Q240: Number of passes that are to be made between points 1 and 4 and between points 2 and 3.

▶ FEED RATE FOR MILLING Q207: Traversing speed of the tool in mm/min when milling the first pass. The TNC calculates the feed rate for all subsequent passes dependent of the stepover factor of the tool (offset less than tool radius = higher feed rate, high stepover factor = lower feed rate)
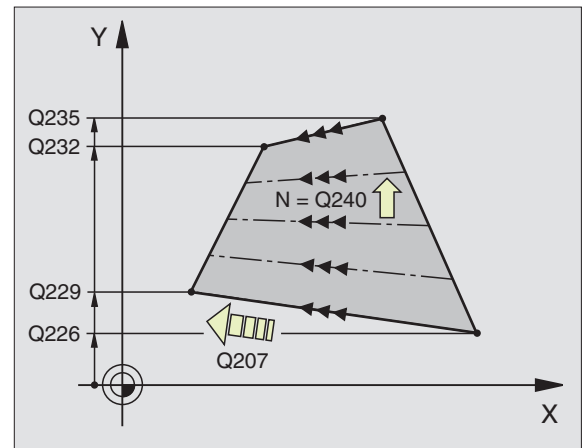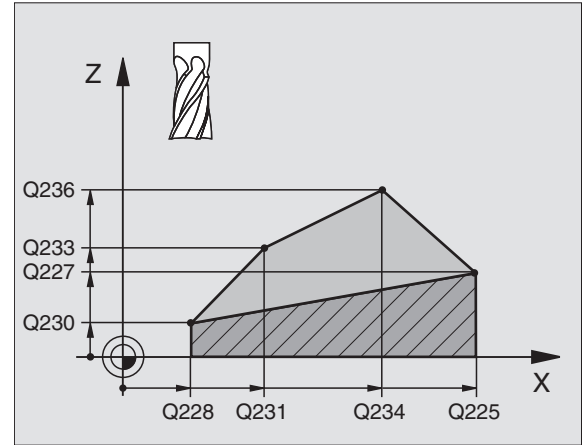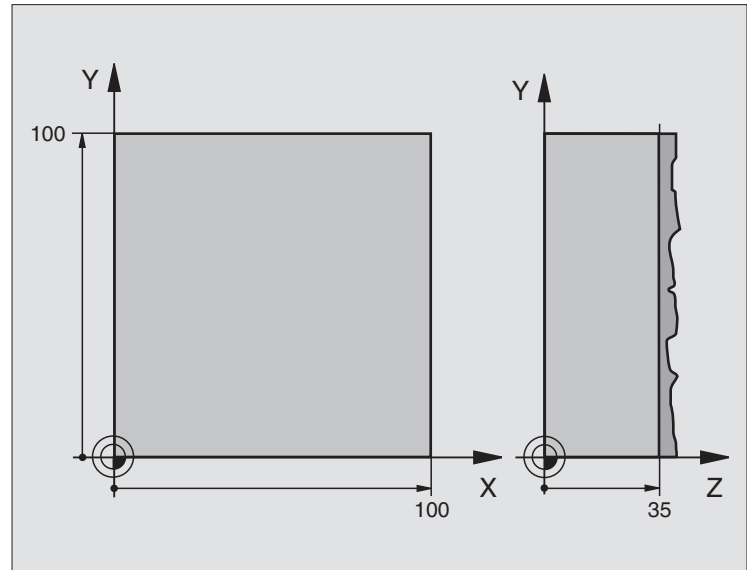
```
0   BEGIN PGM C230 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z+0            Define workpiece blank
2   BLK FORM 0.2 X+100 Y+100 Z+40
3   TOOL DEF 1 L+0 R+5                    Define tool
4   TOOL CALL Q1 Z S3500                  Call tool
5   L Z+250 R0 F MAX                      Retract tool
6   CYCL DEF 230 MULTIPASS MILLNG         Cycle definition: MULTIPASS MILLING
      Q225=+0    ;STARTNG PNT 1ST AXIS
      Q226=+0    ;STARTNG PNT 2ND AXIS
      Q227=+35   ;STARTNG PNT 3RD AXIS
      Q218=100   ;FIRST SIDE LENGTH
      Q219=100   ;SECOND SIDE LENGTH
      Q240=25    ;NUMBER OF CUTS
      Q206=250   ;FEED RATE FOR PLNGNG
      Q207=400   ;FEED RATE FOR MILLNG
      Q209=150   ;STEPOVER FEED RATE
      Q200=2     ;SET-UP CLEARANCE
7   L X+130 Y+0 R0 F MAX M3               Pre-position near the starting point
8   CYCL CALL                             Cycle call
9   L Z+250 R0 F MAX M2                   Retract tool, end of program
10  END PGM C230 MM
```

8.7 Coordinate Transformation Cycles

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

Cycle	Soft key
7 DATUM SHIFT For shifting contours directly within the program or with datum tables	
8 MIRROR IMAGE For mirroring contours	
10 ROTATION For rotating contours in the working plane	
11 SCALING FACTOR For increasing or reducing the size of contours	
26 AXIS-SPECIFIC SCALING For increasing or reducing the size of contours with axis-specific scaling factors	
19 WORKING PLANE For executing machining operations in a tilted coordinate system on machines with swivel heads and/or tilting tables	

Effect of coordinate transformations
A coordinate transformation becomes effective as soon as it is defined — it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:
- Define cycles for basic behavior with a new value, such as scaling factor 1.0

- Execute a miscellaneous function M02, M30, or an END PGM block (depending on machine parameter 7300)

- Select a new program

DATUM SHIFT (Cycle 7)

A datum shift allows machining operations to be repeated at various locations on the workpiece.

Activation

When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display.

> DATUM SHIFT: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid — this can be a datum which has already been shifted.

Cancellation

A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Graphics

If you program a new BLK FORM after a datum shift, you can use machine parameter 7310 to determine whether the BLK FORM is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several parts are machined.
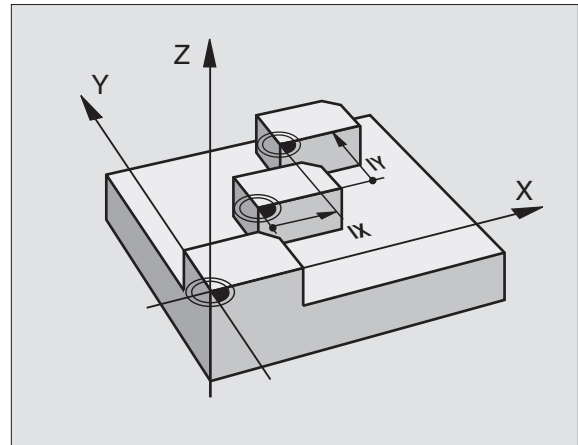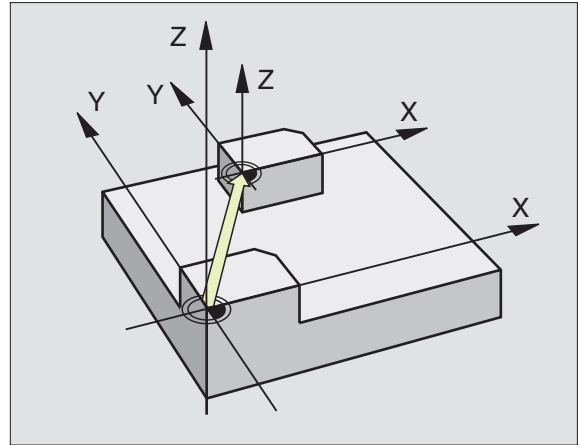
DATUM SHIFT with datum tables (Cycle 7)

☞ If you are using the interactive programming graphics with the datum tables, you must select the desired datum table (status S) in the TEST RUN mode of operation before starting the programming graphics.

If you are working with only one datum table, be sure to activate the correct datum in the program run modes of operation.

Datums from a datum table can be referenced either to the current datum or to the machine datum (depending on machine parameter 7475).

New lines can only be inserted at the end of the table.

The datum points from datum tables are only effective with absolute coordinate values.

Application
Datum tables are applied for

- frequently recurring machining sequences at various locations on the workpiece
- frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

▶ DATUM SHIFT: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number found in the Q parameter.

Cancellation
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.

- Execute a datum shift to the coordinates X=0; Y=0 etc. directly via cycle definition.
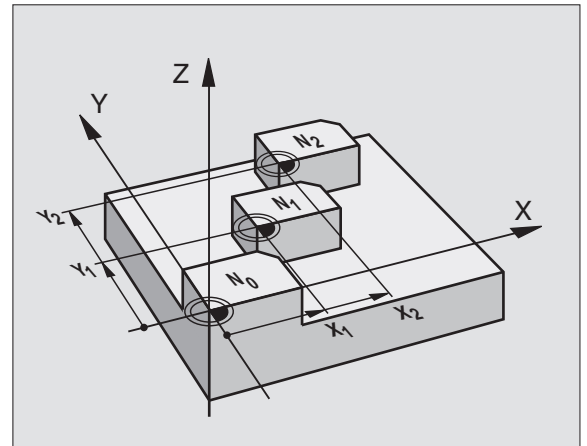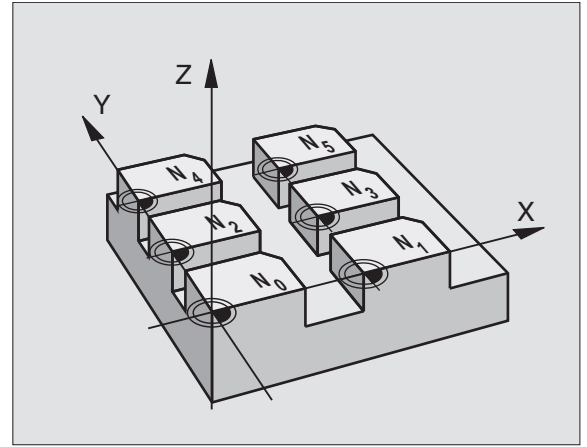
Editing a datum table

Select the datum table in the PROGRAMMING AND EDITING mode of operation.

PGM MGT ▶ To call the file manager, press the PGM MGT key — see section 4.2 "File Management" for more information.

▶ To show datum tables, press the SELECT TYPE and SHOW .D soft keys.

▶ Select the desired table or enter a new file name.

▶ Edit the file. The soft-key row comprises the following functions for editing:

Function	Soft key
Go to beginning of the datum table	BEGIN TABLE
Go to the end of the datum table	END TABLE
Go to the previous page	PAGE ⇧
Go to the next page	PAGE ⇩
Insert line (only possible at the end of table)	INSERT LINE
Delete line	DELETE LINE
Confirm the entered line and go to the beginning of the next line	NEXT LINE

To leave a datum table

Select a different type of file in file management and choose the desired file.

```
MANUAL        DATUM TABLE EDITING
OPERATION     DATUM SHIFT ?

   FILE: NULLTAB          MM
D    X           Y         Z        C       B
0    +0          +0        +0       +0      +0
1    +25         +25       +0       +0      +0
2    +0          +50       +2.5     +0      +0
3    +0          +0        +0       +90     +0
4    +27.25      +0        -3.5     +0      +0
5    +250        +250      +0       +0      +0
6    +350        +350      +10.2    +0      +0
7    +1200       +0        +0       +0      +0
8    +1700       +1200     -25      +0      +0
9    -1700       -1200     +25      +0      +0
10   +0          +0        +0       +0      +0
11   +0          +0        +0       +0      +0
12   +0          +0        +0       +0      +0

BEGIN   END    PAGE    PAGE    INSERT  DELETE  NEXT
TABLE   TABLE   ⇩       ⇧      LINE    LINE    LINE
```

MIRROR IMAGE (Cycle 8)

The TNC can machine the mirror image of a contour in the working plane — see figure at upper right.

Activation

The MIRROR IMAGE cycle becomes effective as soon as it is defined in the program. Cycle 8 is also effective in the POSITION-ING WITH MDI mode of operation. The active mirrored axes are shown in the additional status display.

■ If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).

■ If you mirror two axes, the machining direction remains the same.

The result of the mirror image depends on the location of the datum:

■ If the datum lies on the contour to be mirrored, the element simply flips over — see figure at lower right.

■ If the datum lies outside the contour to be mirrored, the element also "jumps" to another location — see figure at lower right.

▶ MIRROR IMAGE: Enter the axis you wish to mirror. The tool axis cannot be mirrored.

Cancellation

Program the MIRROR IMAGE cycle once again with NO ENT.
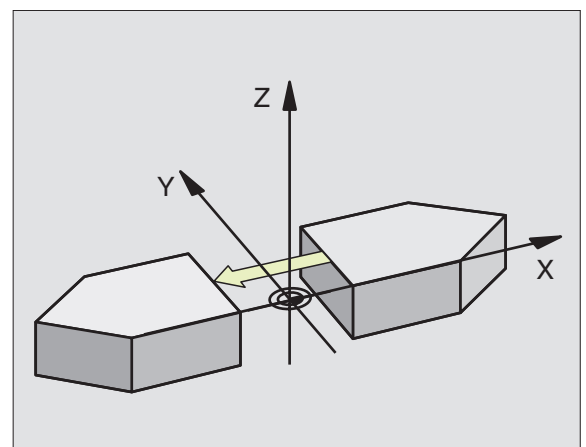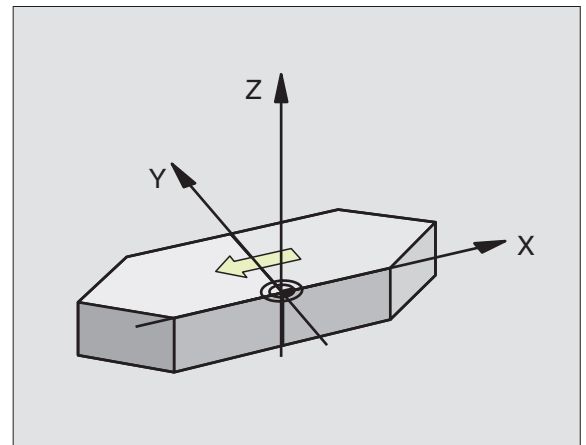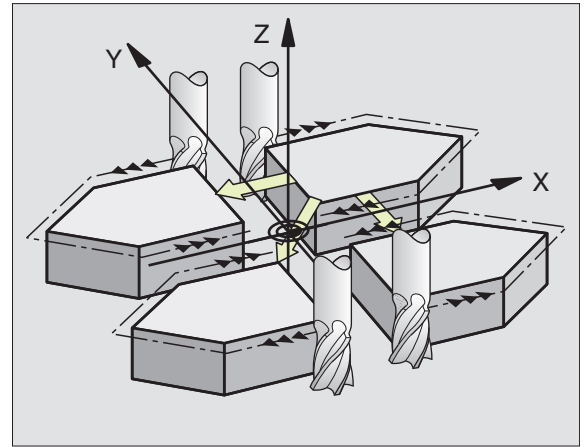
ROTATION (Cycle 10)

The TNC can rotate the coordinate system about the active datum in the working plane within a program.

Activation
The ROTATION cycle becomes effective as soon as it is defined in the program. Cycle 10 is also effective in the POSITIONING WITH MDI mode of operation. The active rotation angle is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane   X axis
- Y/Z plane   Y axis
- Z/X plane   Tool axis

☞ **Before programming, note the following:**

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

After defining Cycle 10, you must move all axes to activate rotation for all axes.

▶ ROTATION: Enter the rotation angle in degrees (°). Input range: –360° to +360° (absolute or incremental).

Cancellation
Program the ROTATION cycle once again with a rotation angle of 0°.
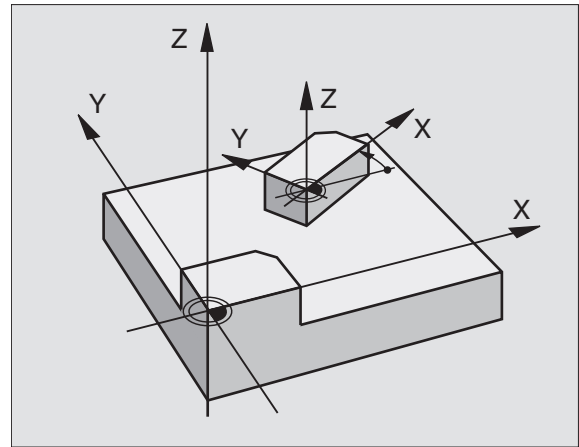
SCALING FACTOR (Cycle 11)

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

Activation

The SCALING FACTOR cycle becomes effective as soon as it is defined in the program. Cycle 11 is also effective in the POSITIONING WITH MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor can be applied

- in the working plane, or on all three coordinate axes at the same time (depending on machine parameter 7410)

- to the dimensions in cycles

- to the parallel axes U,V,W

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

> SCALING FACTOR ?: Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Activation" above)
>
> Enlargement: SCL greater than 1 (up to 99.999 999)
>
> Reduction: SCL less than 1 (down to 0.000 001)

Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1.

You can also enter scaling factors for axis-specific scaling (see Cycle 26).
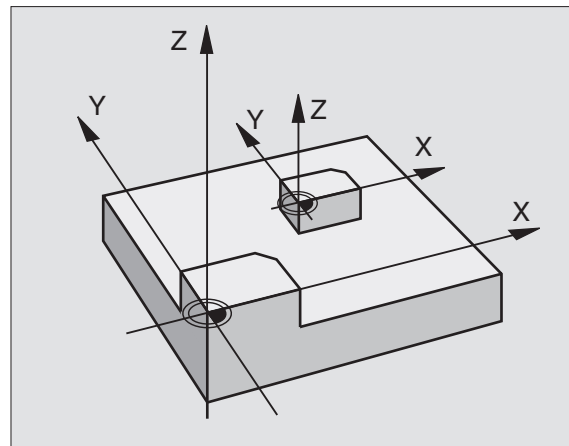
AXIS-SPECIFIC SCALING (Cycle 26)

☞ **Before programming, note the following:**

Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING FACTOR) with reference to the active datum.

Activation

The SCALING FACTOR becomes effective as soon as it is defined in the program. Cycle 26 is also effective in the POSITIONING WITH MDI mode of operation. The active scaling factor is shown in the additional status display.

▶ AXIS AND SCALING FACTOR: Enter the coordinate axis/axes as well as the factor(s) involved in enlarging or reducing. Enter a positive value up to 99.999 999.

▶ CENTER COORDINATES: Enter the center of the axis-specific enlargement or reduction.

The coordinate axes are selected with soft keys.

Cancellation

Program the AXIS-SPECIFIC SCALING cycle once again with a scaling factor of 1 for the same axis.

Example

Axis-specific scaling factors in the working plane

Preset: Quadrangle (see figure at lower right)

Corner 1: X = 20.0 mm   Y = 2.5 mm
Corner 2: X = 32.5 mm   Y = 15.0 mm
Corner 3: X = 20.0 mm   Y = 27.5 mm
Corner 4: X = 7.5 mm   Y = 15.0 mm

■ Enlarge X axis by factor 1.4

■ Reduce Y axis by factor 0.6

■ Center at CCX = 15 mm CCY = 20 mm

Resulting NC blocks

```
CYCL DEF 26.0 AXIS-SPEC. SCALING
CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```
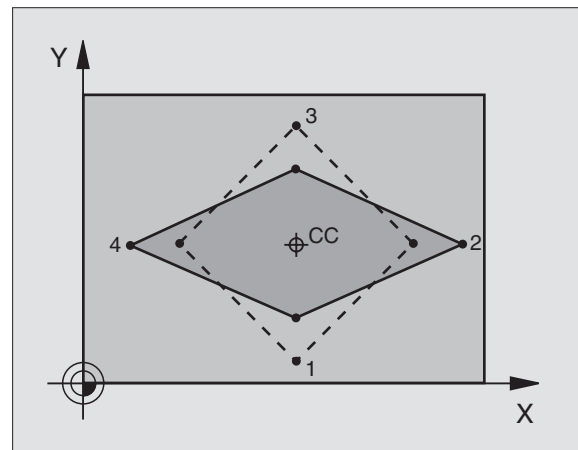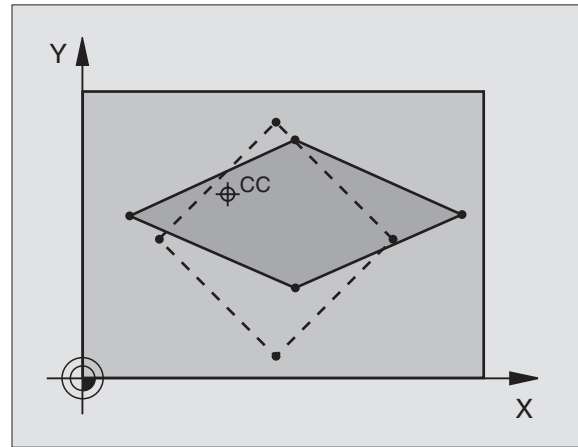
WORKING PLANE (Cycle 19)

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as solid angles. Your machine manual provides more detailed information on this subject.

The working plane is always tilted around the active datum. The fundamentals of this TNC function are described in section 2.5 "Tilting the Working Plane." It is important that you read through this section thoroughly.

Activation

In Cycle 19 you define the position of the working plane by entering tilt angles. The entered angles describe either the direct position of the tilt axes (see figure to the upper right) or the angular components of a space vector (depending on machine parameters, see figure at lower right).

If you program the angular components of the space vector, the TNC automatically calculates the angular position of the tilt axes. The position of the space vector — that is the position of the tool axis — is calculated by the TNC by rotating the **machine-based** coordinate system. The axes are always rotated in the same sequence for calculating the space vector: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You have to move all axes to activate compensation for all axes.

If you set the function TILTING PROGRAM RUN to ACTIVE in the MANUAL OPERATION mode (see section 2.5 "Tilting the Working Plane"), the angular value entered in this menu is overwritten by Cycle 19 WORKING PLANE.

▶ TILT AXIS AND TILT ANGLE: The tilted axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys.

Cancellation

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again by answering the dialog question with the NO ENT key to disable the function.
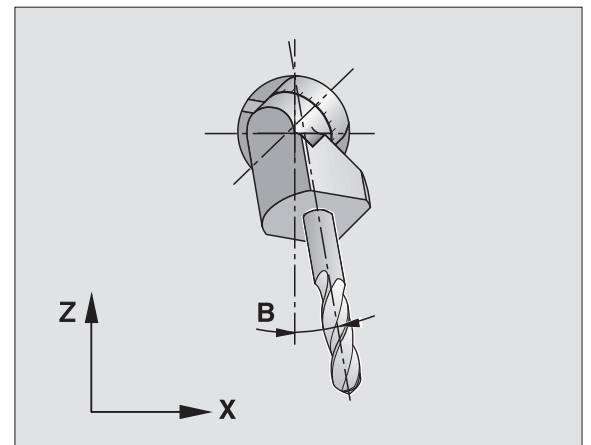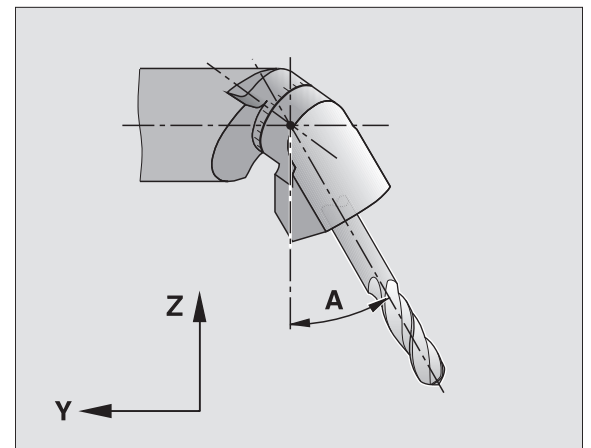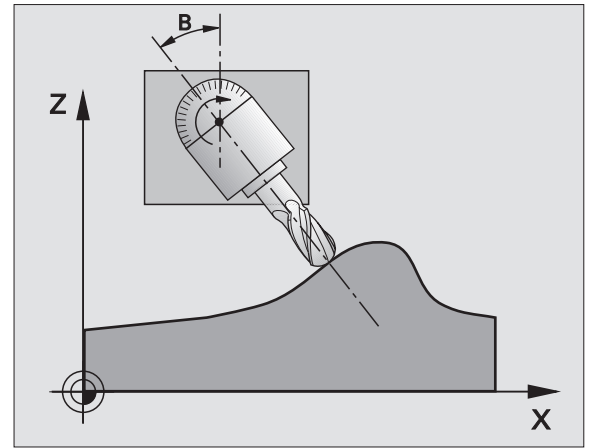
Positioning an axis of rotation

> The machine tool builder determines whether Cycle 19 positions the axes of rotation automatically or whether they must be pre-positioned in the program. Your machine manual provides more detailed information.

If the axes are positioned automatically in Cycle 19:

- The TNC can position only controlled axes

- You can use only preset tools (with the full tool length defined in the TOOL DEF block or in the tool table)

- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting

- The TNC tilts the working plane at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

If the axes are not positioned automatically in Cycle 19, position them before defining the cycle, for example with an L block:

Resulting NC blocks

`L Z+100 R0 FMAX`	
`L X+25 Y+10 R0 FMAX`	
`L A+15 R0 F1000`	Position the axis of rotation
`CYCL DEF 19.0 WORKING PLANE`	Define the angle for calculation of the compensation
`CYCL DEF 19.1 A+15`	
`L Z+80 R0 FMAX`	Activate compensation for the tool axis
`L X-7.5 Y-10 R0 FMAX`	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle 19, the displayed positions (ACTL. and NOML.) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition may not be the same as the coordinates of the last programmed position before Cycle 19.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If any of the software limit switches is traversed the TNC will display an error message.

8.7 Coordinate Transformation Cycles

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle 19. In this case, you are shifting the "machine-based coordinate system."

If you program a datum shift after having activated Cycle 19, you are shifting the "tilted coordinate system."

Important: When resetting, you must use the reverse procedure — the cycle that was last defined should be reset first:

1. Activate datum shift
2. Activate tilting function
3. Activate rotation
...
Machining
...
1. Reset rotation
2. Reset tilting function
3. Reset datum shift

Automatic workpiece measurement in the tilted system

The cycle TCH PROBE 1.0 REF. PLANE enables you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).

Procedure for working with Cycle 19 WORKING PLANE

1 Create program
■ Define the tool (not required, when TOOL.T is active), and enter the full tool length.

■ Call the tool.

■ Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece (clamping devices) during tilting.

■ If required, position the tilt axis or axes with an L block to the appropriate angular value(s) (depending on a machine parameter).

■ Activate datum shift if required.

■ Define Cycle 19 WORKING PLANE; enter the angular values for the tilt axes.

■ Traverse all main axes (X, Y, Z) to activate compensation.

■ Write the program as if the machining process were to be executed in a non-tilted plane.

■ Reset Cycle 19 WORKING PLANE; program 0° for all tilt axes.

■ Disable the WORKING PLANE function; redefine Cycle 19 and answer the dialog question with "NO ENT."

■ Reset datum shift if required.

■ Position the tilt axes to the 0° position if required.

2 Clamp workpiece

3 Preparations in the operating mode POSITIONING WITH MDI
Pre-position the tilt axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.

4 Preparations in the operating mode MANUAL OPERATION
Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the MANUAL OPERATION mode. Enter the angular values for the tilt axes into the menu if the axes are not controlled.

If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the tilted axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Set datum
■ Manually by touching the workpiece with the tool in the non-tilted coordinate system (see section 2.4 "Setting the Datum Without a 3-D Touch Probe")

■ Automatically by using a HEIDENHAIN 3-D touch probe (see section 12.3 "Setting the Datum with a 3-D Touch Probe")

6 Start part program in the operating mode PROGRAM RUN/FULL SEQUENCE

7 MANUAL OPERATION
Use the 3D-ROT soft key to set the function TILT WORKING PLANE to INACTIVE. Enter an angular value of 0° for each axis in the menu (see section 2.5 "Tilting the Working Plane").

Example: Coordinate transformation cycles

Process
- Program the coordinate transformations in the main program
- Program the machining operation in subprogram 1 (see section 9 "Programming: Subprograms and Program Section Repeats")
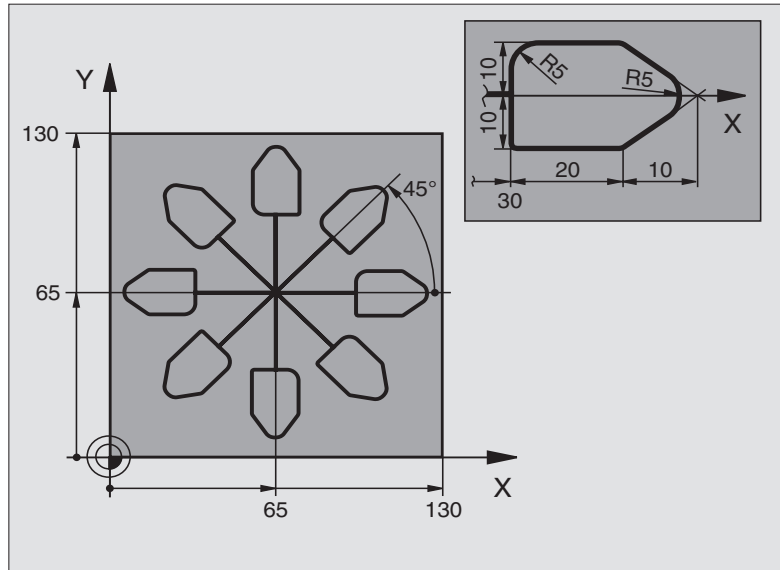
0	BEGIN PGM KOUMR MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define workpiece blank
2	BLK FORM 0.2 X+130 Y+130 Z+0	
3	TOOL DEF 1 L+0 R+1	Define tool
4	TOOL CALL 1 Z S4500	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
7	CYCL DEF 7.1 X+65	
8	CYCL DEF 7.2 Y+65	
9	CALL LBL 1	Call milling operation
10	LBL 10	Set label for program section repeat
11	CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
12	CYCL DEF 10.1 IROT+45	
13	CALL LBL 1	Call milling operation
14	CALL LBL 10 REP 7/7	Return jump to LBL 10; execute the milling operation seven times
15	CYCL DEF 10.0 ROTATION	Cancel rotation
16	CYCL DEF 10.1 ROT+0	
17	CYCL DEF 7.0 DATUM SHIFT	Cancel datum shift
18	CYCL DEF 7.1 X+0	
19	CYCL DEF 7.2 Y+0	
20	L Z+250 R0 F MAX M2	Retract tool, end of program

21	LBL 1	Subprogram 1:
22	L X+0 Y+0 R0 F MAX	Define milling operation
23	L Z+2 R0 F MAX	
24	L Z-5 R0 F200	
25	L X+30 RL	
26	L IY+10	
27	RND R5	
28	L IX+20	
29	L IX+10 IY-10	
30	RND R5	
31	L IX-10 IY-10	
32	L IX-20	
33	L IY+10	
34	L X+0 Y+0 R0 F500	
35	L Z+20 R0 F MAX	
36	LBL 0	
37	END PGM KOUMR MM	

8.8 Special Cycles

DWELL TIME (Cycle 9)

This cycle causes the execution of the next block within a running program to be delayed by the programmed dwell time. A dwell time can be used for such purposes as chip breaking.

Activation
Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.

> DWELL TIME IN SECONDS: Enter the dwell time in seconds
>
> Input range: 0 to 30 000 seconds (approx. 8.3 hours) in increments of 0.001 seconds
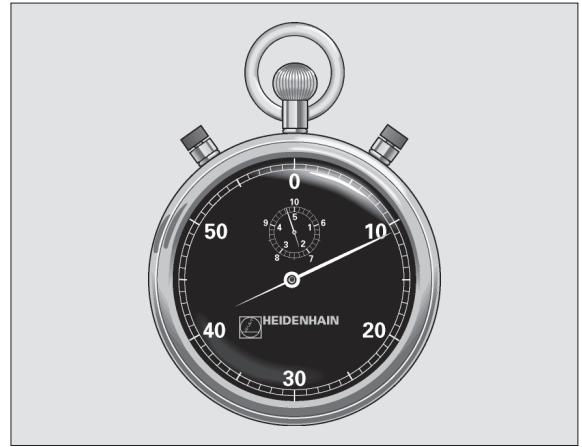
PROGRAM CALL (Cycle 12)

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs and then called like fixed cycles.

☞ **Before programming, note the following:**

If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only to enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path (for example \CONV35\FK1\50.H).

If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.

> PROGRAM NAME: Enter the name of the program you want to call and, if necessary, the directory it is located in.
>
> The program is called with
> ■ CYCL CALL (separate block) or
> ■ M99 (blockwise) or
> ■ M89 (modally)
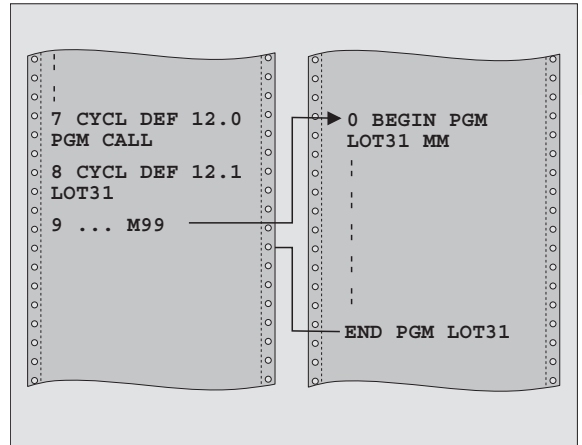
Example: Program call
A callable program 50 is to be called into a program via a cycle call.

Resulting NC blocks

55 CYCL DEF 12.0 PGM CALL	Definition:
56 CYCL DEF 12.1 PGM \KLAR35\FK1\50.H	"Program 50 is a cycle"
57 L X+20 Y+50 FMAX M99	Call program 50

```
7 CYCL DEF 12.0        →  0 BEGIN PGM
PGM CALL                  LOT31 MM
8 CYCL DEF 12.1
LOT31
9 ... M99  ─────────

                          END PGM LOT31
```

ORIENTED SPINDLE STOP (Cycle 13)

> The TNC and the machine tool must be specially pre-pared by the machine tool builder for the use of Cycle 13.

The control can address the machine tool spindle as a 6th axis and rotate it to a given angular position.

Oriented spindle stops are required for

■ Tool changing systems with a defined tool change position

■ Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

Activation
The angle of orientation defined in the cycle is positioned to by entering M19.

If you program M19 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set in a machine parameter.

▶ ANGLE OF ORIENTATION: Enter the angle according to the reference axis of the working plane.

Input range: 0 to 360°

Input resolution: 0.1°

9
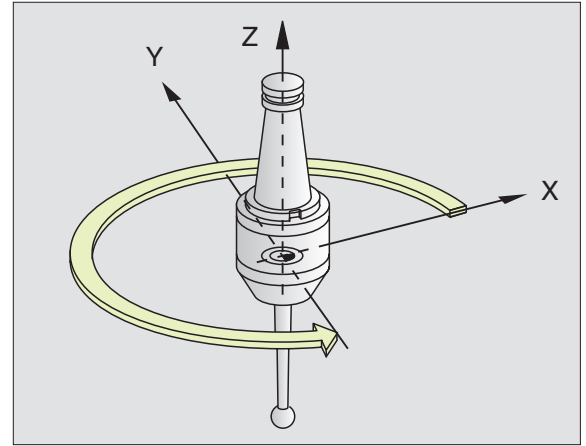
Programming:

Subprograms and
Program Section Repeats

9.1 Marking Subprograms and Program Section Repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

Labels

The beginnings of subprograms and program section repeats are marked in a part program by labels.

A label is identified by a number between 1 and 254. Each label can be set only once with LABEL SET in a program.

LABEL 0 (LBL 0) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

9.2 Subprograms

Operating sequence

1 The TNC executes the part program up to the block in which a subprogram is called with CALL LBL.

2 The subprogram is then executed from beginning to end. The subprogram end is marked with LBL 0.

3 The TNC then resumes the part program from the block after the subprogram call.

Operating limitations

■ A main program can contain up to 254 subprograms.

■ You can call subprograms in any sequence and as often as desired.

■ A subprogram cannot call itself.

■ Write subprograms at the end of the main program (behind the block with M2 or M30).

■ If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

```
0 BEGIN PGM ...
      ①
   CALL LBL1
      ③
   L Z+100 M2
   LBL1
      ②
   LBL0
   END PGM ...
```
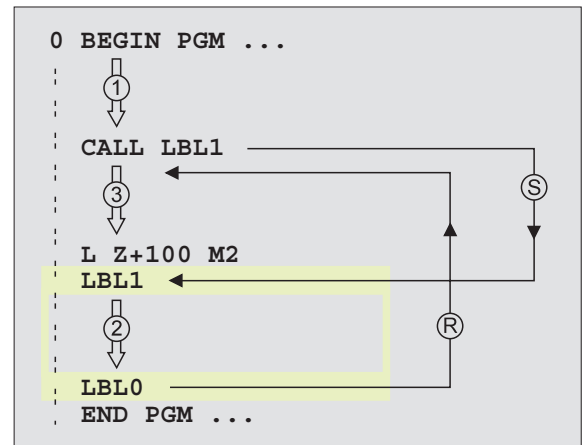
Programming a subprogram

▶ To mark the beginning, press the LBL SET key and enter a LABEL NUMBER.

▶ Enter the subprogram.

▶ To mark the end, press the LBL SET key and enter the LABEL NUMBER "0".

Calling a subprogram

LBL
CALL

▶ To call a subprogram, press the LBL CALL key.

▶ LABEL NUMBER: Enter the label number of the program you wish to call.

▶ REPEAT REP: Ignore the dialog question with the NO ENT key. REPEAT REP is only programmed for program section repeats.

☞ CALL LBL 0 is not permitted (label 0 is only used to mark the end of a subprogram).

9.3 Program Section Repeats

The beginning of a program section repeat is marked by the label LBL. The end of a program section repeat is identified by CALL LBL /REP.

Operating sequence
1 The TNC executes the part program up to the end of the labeled program section, i.e. to the block with CALL LBL /REP.

2 Then the program section between the called LBL and the label call is repeated the number of times entered after REP.

3 The TNC then resumes the part program after the last repetition.

Programming notes
■ You can repeat a program section up to 65 534 times in succession.

■ The number behind the slash after REP indicates the number of repetitions remaining to be run.

■ The total number of times the program section is executed is always one more than the programmed number of repeats.

```
0 BEGIN PGM ...

        1

  LBL1

        2    R   2/1   R   2/0

  CALL LBL1 REP 2/2

        3

  END PGM ...
```
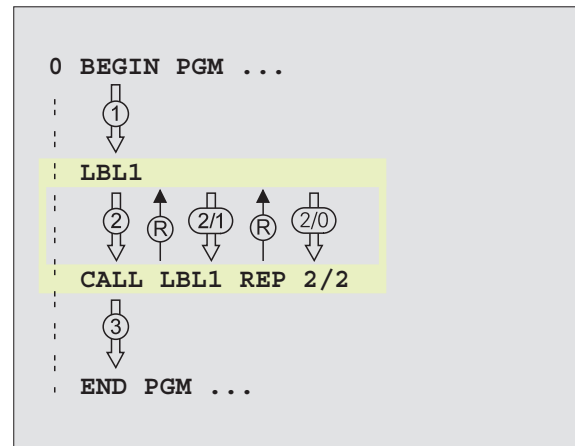
Programming a program section repeat

`LBL SET` ▶ To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat.

▶ Enter the program section.

Calling a program section repeat

`LBL CALL` ▶ Press the LBL CALL key and enter the LABEL NUMBER of the program section you want to repeat as well as the number of repeats (with REPEAT REP).

9.4 Program as Subprogram

1 The TNC executes the part program up to the block in which another program is called with CALL PGM.

2 Then the other program is run from beginning to end.

3 The TNC then resumes the first (calling) part program with the block behind the program call.

Operating limitations

■ No labels are needed to call any program as a subprogram.

■ The called program must contain the miscellaneous functions M2 or M30.

■ The called program must not contain a program call into the calling program.

Calling any program as a subprogram

`PGM CALL` ▶ To call the program, press the PGM CALL key and enter the PROGRAM NAME of the program you wish to call.

☞ You need only to enter the program name if the program you want to call is located in the same directory as the program you are calling it from.

If the program you want to call is not located in the same directory as the program you are calling it from, you must enter the complete path (e.g. \VZW35\ROUGH\PGM1.H).

If you want to call an ISO program, enter the file type .I after the program name.

You can also call a program with Cycle 12 PGM CALL.
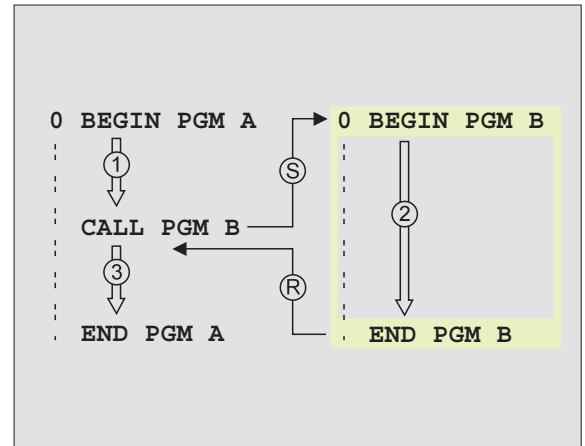
9.5 Nesting

You can nest subprograms and program section repeats in the following ways:

■ Subprograms within a subprogram

■ Program section repeats within a program section repeat

■ Subprograms repeated

■ Program section repeats within a subprogram

Nesting depth
The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

■ Maximum nesting depth for subprograms: 8

■ Maximum nesting depth for calling main programs: 4

■ You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

`0  BEGIN PGM UPGMS MM`	
`...`	
`17  CALL LBL 1`	Call the subprogram marked with LBL1
`...`	
`35  L Z+100 R0 FMAX M2`	Last program block of the main program (with M2)
`36  LBL 1`	Beginning of subprogram 1
`...`	
`39  CALL LBL 2`	Call the subprogram marked with LBL2
`...`	
`45  LBL 0`	End of subprogram 1
`46  LBL 2`	Beginning of subprogram 2
`...`	
`62  LBL 0`	End of subprogram 2
`63  END PGM UPGMS MM`	

Program execution

1st step: Main program UPGMS is executed up to block 17.

2nd step: Subprogram 1 is called, and executed up to block 39.

3rd step: Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called.

4th step: Subprogram 1 is called, and executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program UPGMS.

5th step: Main program UPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

`0  BEGIN PGM REPS MM`	
`...`	
`15   LBL 1`	Beginning of program section repeat 1
`...`	
`20   LBL 2`	Beginning of program section repeat 2
`...`	
`27   CALL LBL 2 REP 2/2`	The program section between this block and LBL 2
`...`	(block 20) is repeated twice.
`35   CALL LBL 1 REP 1/1`	The program section between this block and LBL 1
`...`	(block 15) is repeated once.
`50   END PGM REPS MM`	

Program execution

1st step: Main program REPS is executed up to block 27.

2nd step: Program section between block 27 and block 20 is repeated twice.

3rd step: Main program REPS is executed from block 28 to block 35.

4th step: Program section between block 35 and block 15 is repeated once (including the program section repeat between 20 and block 27).

5th step: Main program REPS is executed from block 36 to block 50 (end of program).

Repeating a subprogram

Example NC blocks

`0 BEGIN PGM UPGREP MM`	
`...`	
`10  LBL 1`	Beginning of the program section repeat
`11  CALL LBL 2`	Subprogram call
`12  CALL LBL 1 REP 2/2`	The program section between this block and LBL1
`...`	(block 10) is repeated twice
`19  L Z+100 R0 FMAX M2`	Last program block of the main program with M2
`20  LBL 2`	Beginning of subprogram
`...`	
`28  LBL 0`	End of subprogram
`29  END PGM UPGREP MM`	

Program execution

1st step: Main program UPGREP is executed up to block 11.

2nd step: Subprogram 2 is called and executed.

3rd step: Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.

4th step: Main program UPGREP is executed from block 13 to block 19. End of program.

Example: Milling a contour in several infeeds

Process
- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Mill the contour
- Repeat downfeed and contour-milling
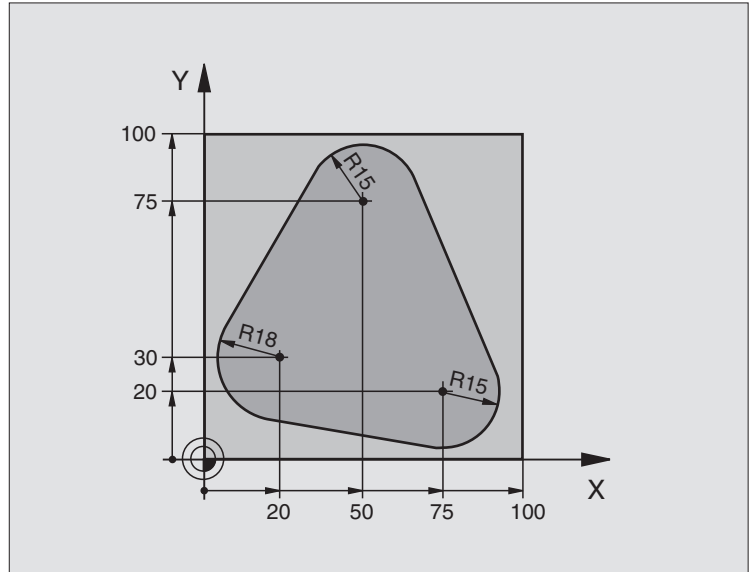
0	BEGIN PGM PGMWDH MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define tool
4	TOOL CALL 1 Z S500	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	L X-20 Y+30 R0 F MAX	Pre-position in the working plane
7	L Z+0 R0 F MAX M3	Pre-position to the workpiece surface
8	LBL 1	Set label for program section repeat
9	L IZ-4 R0 F MAX	Infeed depth in incremental values (in the open)
10	APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach contour
11	FC DR- R18 CLSD+ CCX+20 CCY+30	Contour
12	FLT	
13	FCT DR- R15 CCX+50 CCY+75	
14	FLT	
15	FCT DR- R15 CCX+75 CCY+20	
16	FLT	
17	FCT DR- R18 CLSD- CCX+20 CCY+30	
18	DEP CT CCA90 R+5 F1000	Depart contour
19	L X-20 Y+0 R0 F MAX	Retract tool
20	CALL LBL 1 REP 4/4	Return jump to LBL 1; section is repeated a total of 4 times
21	L Z+250 R0 F MAX M2	Retract tool, end of program
22	END PGM PGMWDH MM	

Example: Groups of holes

Process
- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1
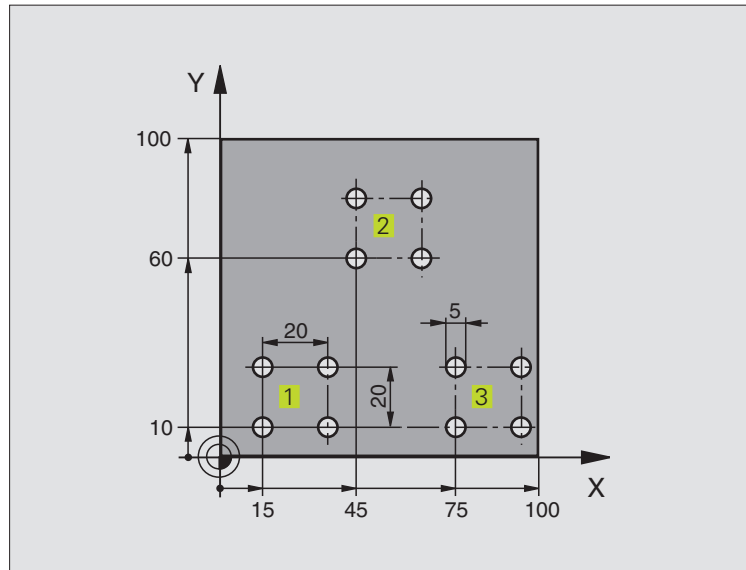
0	BEGIN PGM UP1 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+2.5	Define tool
4	TOOL CALL 1 Z S5000	Call tool
5	L Z+250 R0 F MAX	Retract tool
6	CYCL DEF 200 DRILLING	Cycle definition: DRILLING
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-10 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q202=5 ;PECKING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
7	L X+15 Y+10 R0 F MAX M3	Move to starting point for group 1
8	CALL LBL 1	Call the subprogram for the group
9	L X+45 Y+60 R0 F MAX	Move to starting point for group 2
10	CALL LBL 1	Call the subprogram for the group
11	L X+75 Y+10 R0 F MAX	Move to starting point for group 3
12	CALL LBL 1	Call the subprogram for the group
13	L Z+250 R0 F MAX M2	End of main program

14	LBL 1	Beginning of subprogram 1: Group of holes
15	CYCL CALL	1st hole
16	L IX+20 R0 F MAX M99	Move to 2nd hole, call cycle
17	L IY+20 R0 F MAX M99	Move to 3rd hole, call cycle
18	L IX-20 R0 F MAX M99	Move to 4th hole, call cycle
19	LBL 0	End of subprogram 1
20	END PGM UP1 MM	

Example: Groups of holes with several tools

Process

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2
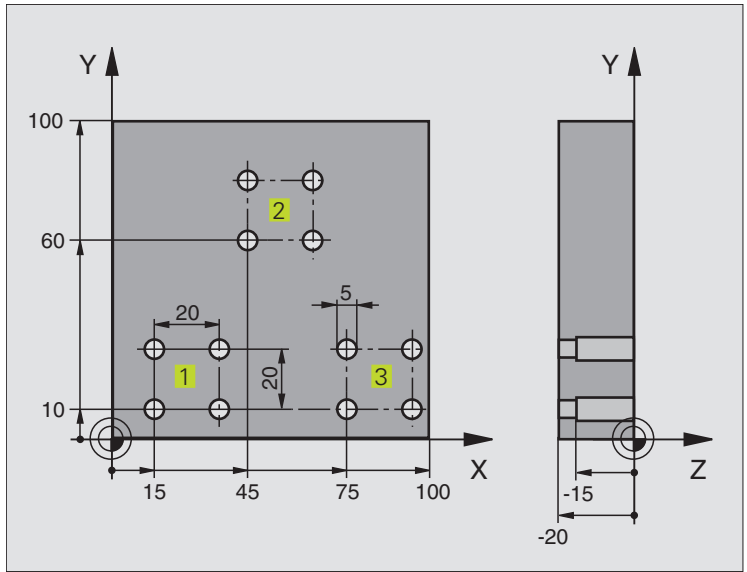
0	BEGIN PGM UP2 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+4	Define tool: center drill
4	TOOL DEF 2 L+0 R+3	Define tool: drill
5	TOOL DEF 3 L+0 R+3.5	Define tool: reamer
6	TOOL CALL 1 Z S5000	Call tool: center drill
7	L Z+250 R0 F MAX	Retract tool

8	CYCL DEF 200 DRILLING	Cycle definition: Centering
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-3 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q202=3 ;PECKING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
9	CALL LBL 1	Call subprogram 1 for the entire hole pattern
10	L Z+250 R0 F MAX M6	Tool change
11	TOOL CALL 2 Z S4000	Tool call: drill
12	FN 0: Q201 = -25	New depth for drilling
13	FN 0: Q202 = +5	New pecking depth for drilling
14	CALL LBL 1	Call subprogram 1 for the entire hole pattern
15	L Z+250 R0 F MAX M6	Tool change
16	TOOL CALL 3 Z S500	Tool call: reamer
17	CYCL DEF 201 REAMING	Cycle definition: REAMING
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q211=0.5 ;DWELL TIME AT DEPTH	
	Q208=400 ;RETRACTION FEED RATE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
18	CALL LBL 1	Call subprogram 1 for the entire hole pattern
19	L Z+250 R0 F MAX M2	End of main program
20	LBL 1	Beginning of subprogram 1: Entire hole pattern
21	L X+15 Y+10 R0 F MAX M3	Move to starting point for group 1
22	CALL LBL 2	Call subprogram 2 for the group
23	L X+45 Y+60 R0 F MAX	Move to starting point for group 2
24	CALL LBL 1	Call subprogram 2 for the group
25	L X+75 Y+10 R0 F MAX	Move to starting point for group 3
26	CALL LBL 2	Call subprogram 2 for the group
27	LBL 0	End of subprogram 1
28	LBL 2	Beginning of subprogram 2: Group of holes
29	CYCL CALL	1st hole with active fixed cycle
30	L IX+20 R0 F MAX M99	Move to 2nd hole, call cycle
31	L IY+20 R0 F MAX M99	Move to 3rd hole, call cycle
32	L IX-20 R0 F MAX M99	Move to 4th hole, call cycle
33	LBL 0	End of subprogram 2
34	END PGM UP2 MM	

10

Programming:

Q Parameters

10.1 Principle and Overview

You can program an entire family of parts on the TNC with a single part program. You do this by entering variables called *Q parameters* instead of fixed numerical values.

Q parameters can represent information such as:

■ Coordinate values

■ Feed rates

■ RPM

■ Cycle data

Q parameters also enable you to program contours that are defined through mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by the letter Q and a number between 0 and 299. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, locally effective only within the program (depending on MP7251)	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs that are stored in the TNC memory	Q200 to Q299

Programming notes
You can mix Q parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between –99 999.9999 and +99 999.9999.

☞ Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius. For further information, see section "10.9 Preassigned Q Parameters."

Calling Q parameter functions
When you are writing a part program, press the "Q" key (below the –/+ key in the keypad for numerical input and axis selection).

The TNC then displays the following soft keys:

Function group	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITH-METIC
Trigonometric functions	TRIGO-NOMETRY
If/then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Enter formula directly from keyboard	FORMULA

10.2 Part Families — Q Parameters in Place of Numerical Values

The Q parameter function FN0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Resulting NC blocks

```
15  FN0:  Q10 = 25      ASSIGN:
...                     Q10 contains the value 25
25  L  X  +Q10          Means  L   X +25
```

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example
Cylinder with Q parameters

Cylinder radius	R	=	Q1
Cylinder height	H	=	Q2

Cylinder Z1   Q1 = +30
                   Q2 = +10

Cylinder Z2   Q1 = +10
                   Q2 = +50

10.3 Describing Contours Through Mathematical Functions

The Q parameters listed below enable you to program basic mathematical functions in a part program:

▶ To select the Q parameter function, press the Q key (at the right, in the keypad for numerical input). The Q parameter functions are displayed in a soft-key row.

▶ To select the mathematical functions, press the BASIC ARITHME-TIC soft key. The TNC then displays the following soft keys:

Function	Soft key
FN0: ASSIGN Example: FN0: Q5 = +60 Assigns a numerical value.	FN0 X = V
FN1: ADDITION Example: FN1: Q1 = –Q2 + –5 Calculates and assigns the sum of two values.	FN1 X + V
FN2: SUBTRACTION Example: FN2: Q1 = +10 – +5 Calculates and assigns the difference of two values.	FN2 X – V
FN3: MULTIPLICATION Example: FN3: Q2 = +3 ∗ +3 Calculates and assigns the product of two values.	FN3 X ∗ V
FN4: DIVISION Example: FN4: Q4 = +8 DIV +Q2 Calculates and assigns the quotient of two values. **Not permitted:** division by 0	FN4 X / V
FN5: SQUARE ROOT Example: FN5: Q20 = SQRT 4 Calculates and assigns the square root of a number. **Not permitted:** square root of a negative number	FN5 SQRT

"Values" in the above table are entered to the right of the "=" character and can be any of the following:

■ Two numbers

■ Two Q parameters

■ A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Example: Programming fundamental operations

| Q | To select Q parameter functions, press the Q key. |

| BASIC ARITH-METIC | To select fundamental operations, press the BASIC ARITHMETIC soft key. |

| FN0 X = Y | To select the Q parameter function ASSIGN, press the FN0 X = Y soft key. |

PARAMETER NUMBER FOR RESULT?

| 5 ENT | Enter a parameter number, for example 5. |

FIRST VALUE / PARAMETER?

| 10 ENT | Assign a value to Q5, for example 10. |

| Q | To select the Q parameter functions, press the Q key. |

| BASIC ARITH-METIC | To select fundamental operations, press the BASIC ARITHMETIC soft key. |

| FN3 X * Y | To select the Q parameter function MULTIPLICATION, press the FN3 X ∗ Y soft key. |

PARAMETER NUMBER FOR RESULT?

| 12 ENT | Enter a Q parameter number, for example 12. |

FIRST VALUE / PARAMETER

| Q 5 ENT | Enter Q5 for the first value. |

SECOND VALUE / PARAMETER

| 7 ENT | Enter 7 for the second value. |

The TNC displays the following program blocks:

```
16 FN0: Q5 = +10
```
```
17 FN3: Q12 = +Q5 * +7
```

10.4 Trigonometric Functions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a right triangle, the trigonometric functions of the angle α are defined by the following equations:

Sine: $\sin \alpha = a / c$

Cosine: $\cos \alpha = b / c$

Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

■ c is the side opposite the right angle

■ a is the side opposite the angle α

■ b is the third side.

The TNC can find the angle from the tangent:

$\alpha = \arctan \alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

  a = 10 mm

  b = 10 mm

  $\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore:

  $a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

  $c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions
Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table at right.

Programming method: Trigonometric functions are programmed similar to fundamental operations, see page 223 "Example: Programming fundamental operations."

Function	Soft key
FN6: SINE Example: FN6: Q20 = SIN–Q5 Calculate the sine of an angle in degrees (°) and assign it to a parameter.	FN6 SIN(X)
FN7: COSINE Example: FN7: Q21 = COS–Q5 Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	FN7 COS(X)
FN8: ROOT-SUM OF SQUARES Example: FN8: Q10 = +5 LEN +4 Take the square root of the sum of two squares and assign it to a parameter.	FN8 X LEN V
FN13: ANGLE Example: FN13: Q20 = +10 ANG–Q1 Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.	FN13 X ANG V

10.5 If-Then Decisions with Q Parameters

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see section 9 "Subprograms and Program Section Repeats"). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter PGM CALL after the block with the target label.

Unconditional jumps
An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN9: IF+10 EQU+10 GOTO LBL1

Programming If-Then decisions
Press the JUMP soft key to call the if-then conditions. The TNC then displays the following soft keys:

Function	Soft key
FN9: IF EQUAL, JUMP Example: FN9: IF +Q1 EQU +Q3 GOTO LBL 5 If the two values or parameters are equal, jump to the given label.	FN9 IF X EQ Y GOTO
FN10: IF NOT EQUAL, JUMP Example: FN10: IF +10 NE –Q5 GOTO LBL 10 If the two values or parameters are not equal, jump to the given label.	FN10 IF X NE Y GOTO
FN11: IF GREATERTHAN, JUMP Example: FN11: IF+Q1 GT+10 GOTO LBL 5 If the first value or parameter is greater than the second value or parameter, jump to the given label.	FN11 IF X GT Y GOTO
FN12: IF LESS THAN, JUMP Example: FN12: IF+Q5 LT+0 GOTO LBL 1 If the first value or parameter is less than the second value or parameter, jump to the given label.	FN12 IF X LT Y GOTO

Abbreviations used:

IF   If

EQU Equals

NE   Not equal

GT   Greater than

LT   Less than

GOTO Go to

10.6 Checking and Changing Q Parameters

During a program run or test run, you can check or change Q parameters if necessary.

▶ If you are in a program run, interrupt it (for example by pressing the machine STOP button and the INTERNAL STOP soft key). If you are doing a test run, interrupt it.

Q

    ▶ To call the Q parameter functions, press the Q key.

    ▶ Enter the Q parameter number and press the ENT key. The TNC displays the current value of the Q parameter in the dialog line.

    ▶ If you wish to change the value, enter a new value, confirm it with the ENT key and conclude your entry with the END key.

    To leave the value unchanged, terminate the dialog with the END key.

```
MANUAL      TEST RUN
OPERATION   Q201 = -50
0   BEGIN PGM 1 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-40
2   BLK FORM 0.2 X+100 Y+100 Z+0
*   - BOHRPLATTE ID-NR 257943KL1
4   TOOL CALL 1 Z S4500
5   L Z+100 R0 F MAX
6   CYCL DEF 203 UNIVERSL DRILLNG
    Q200=2     ;SET-UP CLEARANCE
    Q201=-50   ;DEPTH
    Q206=250   ;FEED RATE FOR PLNGNG
    Q202=0     ;PECKING DEPTH
    Q210=0     ;DWELL TIME AT TOP
    Q203=+0    ;SURFACE COORDINATE
    Q204=100   ;2ND SET-UP CLEARANCE
    Q212=0     ;DECREMENT

                                          END
```

10.7 Additional Functions

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key
FN14:ERROR Display error messages	FN14 ERROR=
FN15:PRINT Unformatted output of texts or Q parameter values	FN15 PRINT
FN16:F-PRINT Formatted output of texts or Q parameter values	FN16 F-PRINT
FN18:SYS-DATUM READ Read system data	FN18 SYS-DATUM READ
FN19:PLC Transfer values to the PLC	FN19 PLC=

FN14: ERROR
Displaying error messages

With the function FN14: ERROR you can call messages under program control. The messages were preprogrammed by the machine tool builder or by HEIDENHAIN. If the TNC encounters a block with FN 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers and the associated texts are listed in the table at right.

Resulting NC block
The TNC is to display the text stored under error number 254:

```
180 FN 14:ERROR = 254
```

Range of error numbers	Standard dialog text
0 ... 299	FN 14: ERROR CODE 0 299
300 ... 999	No standard dialog text prepared
1000 ... 1099	Internal error messages (see table at right)

Error code and text	
1000	SPINDLE MUST BE TURNING
1001	TOOL AXIS IS MISSING
1002	SLOT WIDTH TOO LARGE
1003	TOOL RADIUS TOO LARGE
1004	RANGE EXCEEDED
1005	START POSITION INCORRECT
1006	ROTATION NOT PERMITTED
1007	SCALING FACTOR NOT PERMITTED
1008	MIRRORING NOT PERMITTED
1009	DATUM SHIFT NOT PERMITTED
1010	FEED RATE IS MISSING
1011	ENTRY VALUE INCORRECT
1012	WRONG SIGN PROGRAMMED
1013	ENTERED ANGLE NOT PERMITTED
1014	TOUCH POINT INACCESSIBLE
1015	TOO MANY POINTS
1016	CONTRADICTORY ENTRY
1017	CYCL INCOMPLETE
1018	PLANE WRONGLY DEFINED
1019	WRONG AXIS PROGRAMMED
1020	WRONG RPM
1021	RADIUS COMP. UNDEFINED
1022	ROUNDING-OFF UNDEFINED
1023	ROUNDING RADIUS TOO LARGE
1024	PROGRAM START UNDEFINED
1025	EXCESSIVE SUBPROGRAMMING
1026	ANGLE REFERENCE MISSING

FN15:PRINT
Unformatted output of texts or Q parameter values

☞ Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see section 14 "MOD Functions, Setting the Date Interface").

The function FN15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode).

To output dialog texts and error messages with FN15: PRINT "numerical value"
Numerical values from 0 to 99:   Dialog texts for OEM cycles

Numerical values exceeding 100: PLC error messages

Example: Output of dialog text 20

```
67  FN15:PRINT 20
```

To output dialog texts and error messages with FN15: PRINT "Q parameter"
Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously. The TNC separates them with slashes.

Example: Output of dialog text 1 and numerical value for Q1

```
70  FN15:PRINT 1/Q1
```

FN16:F-PRINT
Formatted output of texts or Q parameter values

☞ Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see Chapter 14 "MOD Functions, Setting the Date Interface").

The function FN16: F-PRINT transfers Q parameter values and texts in a selectable format through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN16RUN.A (output in program run mode) or in the file %FN16SIM.A (output in test run mode).

To output the formatted texts and Q parameter values, create a text file with the TNC's text editor, and define the output format and Q parameters in this file.

```
MANUAL
OPERATION   PROGRAMMING AND EDITING

RS232 INTERFACE        RS422 INTERFACE

MODE OF OP.: LSV-2    MODE OF OP.: LSV-2
BAUD RATE             BAUD RATE
FE   :   9600         FE   :   9600
EXT1 :   9600         EXT1 :   9600
EXT2 :   9600         EXT2 :   9600
LSV-2:   9600         LSV-2:   9600

ASSIGN:

PRINT     :  TNC:\SCREENS\NEUEBA
PRINT-TEST :
```

| | RS 232 RS 422 SETUP | USER PARAMETER | HELP | | | | END |

Example of a text file to define the output format:

"TEST RECORD BUCKET WHEEL GRAVITY CENTER";

"_____";

"NO. OF MEASURED VALUES : = 1";

"***";

"X1 = %4.3LF", Q31;

"Y1 = %4.3LF", Q32;

"Z1 = %2I", Q33;

"* *";

When you create a text file, use the following formatting functions:

Special character	Function
"............."	Define output format for texts and variables between the quotation marks
%5.4LF	Define format for Q parameters (long, floating): 5 places before and 4 places behind the decimal point
%2I	Define format for Q parameters (integer): Integer with 5 places (in this example: with 2 places)
,	Separation character between output format and parameter
;	End of block character

To activate output, program FN16: F-PRINT in the part program:

```
96  FN16:F-PRINT  TNC:\MASK\MASK1.A
```

The TNC then transfers the associated file %FN16SIM.A:

```
CALIBRAT.  CHART   IMPELLER   CENTER   GRAVITY

_____

NO.  OF  MEASURED  VALUES  :  =  1
*******************************************
X1  =  149.360
Y1  =  25.509
Z1  =  37
*******************************************
```

FN18:SYS-DATUM READ
Reading system data

With the function FN18: SYS-DATUM READ you can read system data and store them in Q parameters. You select the system data through a group number (ID number), and additionally through a number and an index.

Group name, ID No.	Number	System data
Program information, 10	1	MM/inch condition
	2	Overlap factor for pocket milling
	3	Number of active fixed cycle
Machine status, 20	1	Active tool number
	2	Prepared tool number
	3	Active tool axis
	4	Programmed spindle rpm
	5	Active spindle status
	8	Coolant status
	9	Active feed rate
Data from the tool table, 50	1	Tool length
	2	Tool radius
	3	Tool radius R2
	4	Oversize for tool length DL
	5	Oversize for tool radius DR
	6	Oversize for tool radius DR2
	7	Tool inhibited (0 or 1)
	8	Number of replacement tool
	9	Maximum tool age TIME1
	10	Maximum tool age TIME2
	11	Current tool age CUR. TIME
	12	PLC status
	13	Maximum tooth length LCUTS
	14	Maximum plunge angle ANGLE
	15	TT: Number of teeth CUT
	16	TT: Wear tolerance for length LTOL
	17	TT: Wear tolerance for radius RTOL
	18	TT: Rotational direction DIRECT (3 or 4)
	19	TT: Offset for radius R-OFFS
	20	TT: Offset for length L-OFFS
	21	TT: Breakage tolerance in length LBREAK
	22	TT: Breakage tolerance in radius RBREAK

Group name	Group No.	Number	Index	System data
Active transformations	210	1	—	Basic rotation in MANUAL OPERATION mode
		2	—	Programmed rotation with Cycle 10
		3	—	Active mirror axis
				0: mirroring not active
				+1: X axis mirrored
				+2: Y axis mirrored
				+4: Z axis mirrored
				+8: IV axis mirrored
				+16: V axis mirrored
				Combinations = sum of individual axes
		4	1	Active scaling factor in X axis
		4	2	Active scaling factor in Y axis
		4	3	Active scaling factor in Z axis
		4	4	Active scaling factor in IV axis
		4	5	Active scaling factor in V axis
		5	1	3D ROT A axis
		5	2	3D ROT B axis
		5	3	3D ROT C axis
Data from the last TOOL CALL block	60	1		Tool number
		2		Tool axis
		3		Spindle rpm
		4		Oversize for tool length DL
		5		Oversize for tool radius DR

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
55  FN18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN19:PLC
Transferring values to the PLC

The function FN19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
56  FN19:PLC=+10/+Q3
```

10.8 Entering Formulas Directly

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Entering formulas
Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
Addition Example: Q10 = Q1 + Q5	+
Subtraction Example: Q25 = Q7 – Q108	–
Multiplication Example: Q12 = 5 ∗ Q5	∗
Division Example: Q25 = Q1 / Q2	/
Open parentheses Example: Q12 = Q1 ∗ (Q2 + Q3)	(
Close parentheses Example: Q12 = Q1 ∗ (Q2 + Q3)	)
Square Example: Q15 = SQ 5	SQ
Square root Example: Q22 = SQRT 25	SQRT
Sine of an angle Example: Q44 = SIN 45	SIN
Cosine of an angle Example: Q45 = COS 45	COS
Tangent of an angle Example: Q46 = TAN 45	TAN

Mathematical function	Soft key
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: Q10 = ASIN 0.75	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: Q11 = ACOS Q40	ACOS
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q50	ATAN
Powers Example: Q15 = 3^3	^
π 3.14159	PI
Natural logarithm (LN) of a number Base 2.7183 Example: Q15 = LN Q11	LN
Logarithm of a number, base 10 Example: Q33 = LOG Q22	LOG
Exponential function, 2.7183^n Example: Q1 = EXP Q12	EXP
Negate (multiplication by -1) Example: Q2 = NEG Q1	NEG
Drop places after the decimal point (form an integer) Example: Q3 = INT Q42	INT
Absolute value Example: Q4 = ABS Q22	ABS
Drop places before the decimal point (form a fraction) Example: Q5 = FRAC Q23	FRAC

Rules for formulas

Mathematical formulas are programmed according to the following rules:

- **Higher-level operations are performed first**
  (multiplication and division before addition and subtraction)

`12 Q1 = 5 * 3 + 2 * 10 = 35`

1st step: $5 * 3 = 15$
2nd step: $2 * 10 = 20$
3rd step: $15 + 20 = 35$

`13 Q2 = SQ 10 − 3^3 = 73`

1st step: $10^2 = 100$
2nd step: $3^3 = 27$
3rd step: $100 − 27 = 73$

- **Distributive law**
  for calculating with parentheses

$a * (b + c) = a * b + a * c$

10.8 Entering Formulas Directly

Programming example

Calculate an angle with arc tangent as opposite side (Q12) and adjacent side (Q13); then store in Q25.

| Q FORMULA | To select the formula entering function, press the Q key and FORMULA soft key. |

PARAMETER NUMBER FOR RESULT?

| 25 ENT | Enter the parameter number. |

| ▷ TAN | Shift the soft-key row and select the arc tangent function. |

| ◁ (| Shift the soft-key row and open parentheses. |

| Q 12 | Enter Q parameter number 12. |

| / | Select division. |

| Q 13 | Enter Q parameter number 13. |

|) END | Close parentheses and conclude formula entry. |

Resulting NC block

`37 Q25 = ATAN (Q12/Q13)`

10.9 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107
The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Tool radius: Q108
The current value of the tool radius is assigned to Q108.

Tool axis: Q109
The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
Z axis	Q109 = 2
Y axis	Q109 = 1
X axis	Q109 = 0

Spindle status: Q110
The value of Q110 depends on which M function was last programmed for the spindle:

M function	Parameter value
No spindle status defined	Q110 = –1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112
The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with PGM CALL) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118
Vth axis	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 110

Actual-nominal deviation	Parameter
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC

Coordinates	Parameter
A axis	Q120
B axis	Q121
C axis	Q122

Example: Ellipse

Process

■ The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating steps you define for the lines, the smoother the curve becomes.

■ The machining direction can be altered by changing the entries for the starting and end angles in the plane:

Clockwise machining direction:
starting angle > end angle
Counterclockwise machining direction:
starting angle < end angle

■ The tool radius is not taken into account.

0	BEGIN PGM ELLIPSE MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q3 = +50	Semiaxis in X
4	FN 0: Q4 = +30	Semiaxis in Y
5	FN 0: Q5 = +0	Starting angle in the plane
6	FN 0: Q6 = +360	End angle in the plane
7	FN 0: Q7 = +40	Number of calculating steps
8	FN 0: Q8 = +0	Rotational position of the ellipse
9	FN 0: Q9 = +5	Milling depth
10	FN 0: Q10 = +100	Pecking depth
11	FN 0: Q11 = +350	Feed rate for milling
12	FN 0: Q12 = +2	Setup clearance for pre-positioning
13	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+2.5	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	L Z+100 R0 F MAX M2	Retract tool, end of program

10.10 Programming Examples

`20   LBL 10`	Subprogram 10: Machining operation
`21   CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center of ellipse
`22   CYCL DEF 7.1 X+Q1`	
`23   CYCL DEF 7.2 Y+Q2`	
`24   CYCL DEF 10.0 ROTATION`	Account for rotational position in the plane
`25   CYCL DEF 10.1 ROT+Q8`	
`26   Q35 = (Q6 - Q5) / Q7`	Calculate angle increment
`27   Q36 = Q5`	Copy starting angle
`28   Q37 = 0`	Set counter
`29   Q21 = Q3 * COS Q36`	Calculate X coordinate for starting point
`30   Q22 = Q4 * SIN Q36`	Calculate Y coordinate for starting point
`31   L X+Q21 Y+Q22 R0 F MAX M3`	Move to starting point in the plane
`32   L Z+Q12 R0 F MAX`	Pre-position in tool axis to setup clearance
`33   L Z-Q9 R0 FQ10`	Move to working depth
`34   LBL 1`	
`35   Q36 = Q36 + Q35`	Update the angle
`36   Q37 = Q37 + 1`	Update the counter
`37   Q21 = Q3 * COS Q36`	Calculate the current X coordinate
`38   Q22 = Q4 * SIN Q36`	Calculate the current Y coordinate
`39   L X+Q21 Y+Q22 R0 FQ11`	Move to next point
`40   FN 12: IF +Q37 LT +Q7 GOTO LBL 1`	Unfinished? If not finished return to LBL 1
`41   CYCL DEF 10.0 ROTATION`	Reset rotation
`42   CYCL DEF 10.1 ROT+0`	
`43   CYCL DEF 7.0 DATUM SHIFT`	Reset datum shift
`44   CYCL DEF 7.1 X+0`	
`45   CYCL DEF 7.2 Y+0`	
`46   L Z+Q12 R0 F MAX`	Move to setup clearance
`47   LBL 0`	End of subprogram
`48   END PGM ELLIPSE MM`	

Example: Concave cylinder machined with spherical cutter

Process

■ This program requires a spherical cutter.

■ The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.

■ The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).

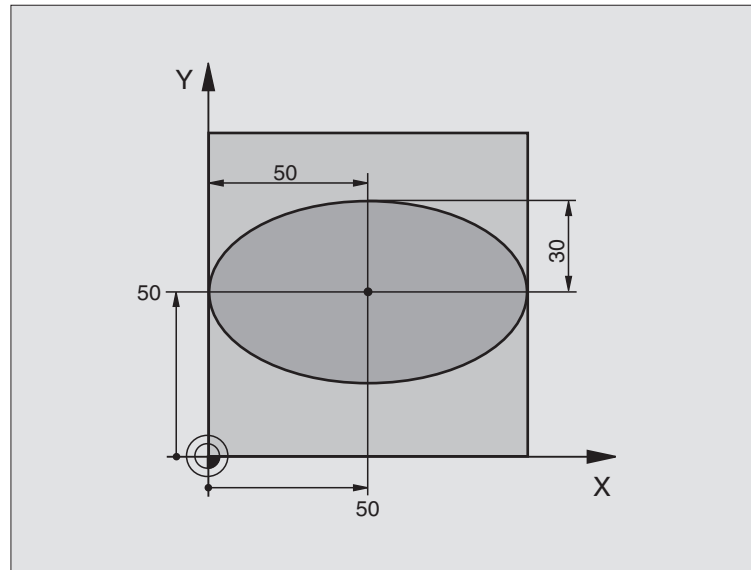■ The machining direction can be altered by changing the entries for the starting and end angles in space:

Clockwise machining direction:
starting angle > end angle
Counterclockwise machining direction:
starting angle < end angle

■ The tool radius is compensated automatically.

0	BEGIN PGM CYLIN MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +0	Center in Y axis
3	FN 0: Q3 = +0	Center in Z axis
4	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
5	FN 0: Q5 = +270	End angle in space (Z/X plane)
6	FN 0: Q6 = +40	Radius of the cylinder
7	FN 0: Q7 = +100	Length of the cylinder
8	FN 0: Q8 = +0	Rotational position in the X/Y plane
9	FN 0: Q10 = +5	Allowance for cylinder radius
10	FN 0: Q11 = +250	Feed rate for pecking
11	FN 0: Q12 = +400	Feed rate for milling
12	FN 0: Q13 = +90	Number of cuts
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+3	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	CALL LBL 10	Call machining operation
21	L Z+100 R0 F MAX M2	Retract tool, end of program

22	`LBL 10`	Subprogram 10: Machining operation
23	`Q16 = Q6 - Q10 - Q108`	Account for allowance and tool, based on the cylinder radius
24	`FN 0: Q20 = +1`	Set counter
25	`FN 0: Q24 = +Q4`	Copy starting angle in space (Z/X plane)
26	`Q25 = (Q5 - Q4) / Q13`	Calculate angle increment
27	`CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center of cylinder (X axis)
28	`CYCL DEF 7.1 X+Q1`	
29	`CYCL DEF 7.2 Y+Q2`	
30	`CYCL DEF 7.3 Z-Q3`	
31	`CYCL DEF 10.0 ROTATION`	Account for rotational position in the plane
32	`CYCL DEF 10.1 ROT+Q8`	
33	`L X+0 Y+0 R0 F MAX`	Pre-position in the plane to the cylinder center
34	`L Z+5 R0 F1000 M3`	Pre-position in the tool axis
35	`CC Z+0 X+0`	Set pole in the Z/X plane
36	`LP PR+Q16 PA+Q24 FQ11`	Move to starting position on cylinder, plunge-cutting obliquely into the material
37	`LBL 1`	
38	`L Y+Q7 R0 FQ11`	Longitudinal cut in Y+ direction
39	`FN 1: Q20 = +Q20 + +1`	Update counter
40	`FN 1: Q24 = +Q24 + +Q25`	Update solid angle
41	`FN 11: IF +Q20 GT +Q13 GOTO LBL 99`	Finished? If finished, jump to end.
42	`LP PR+Q16 PA+Q24 FQ12`	Move in an approximated "arc" for the next longitudinal cut
43	`L Y+0 R0 FQ11`	Longitudinal cut in Y– direction
44	`FN 1: Q24 = +Q24 + +Q25`	Update counter
45	`FN 1: Q20 = +Q20 + +1`	Update solid angle
46	`FN 12: IF +Q20 LT +Q13 GOTO LBL 1`	Unfinished? If not finished, return to LBL 1
47	`LBL 99`	
48	`CYCL DEF 10.0 ROTATION`	Reset rotation
49	`CYCL DEF 10.1 ROT+0`	
50	`CYCL DEF 7.0 DATUM SHIFT`	Reset datum shift
51	`CYCL DEF 7.1 X+0`	
52	`CYCL DEF 7.2 Y+0`	
53	`CYCL DEF 7.3 Z+0`	
54	`LBL 0`	End of subprogram
55	`END PGM CYLIN MM`	

Example: Convex sphere machined with end mill

Process

- This program requires an end mill.

- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.

- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).

- The tool moves upward in three-dimensional cuts.

- The tool radius is compensated automatically.

0	BEGIN PGM BALL MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
4	FN 0: Q5 = +0	End angle in space (Z/X plane)
5	FN 0: Q14 = +5	Angle increment in space
6	FN 0: Q6 = +45	Radius of the sphere
7	FN 0: Q8 = +0	Starting angle of rotational position in the X/Y plane
8	FN 0: Q9 = +360	End angle of rotational position in the X/Y plane
9	FN 0: Q18 = +10	Angle increment in the X/Y plane for roughing
10	FN 0: Q10 = +5	Allowance in sphere radius for roughing
11	FN 0: Q11 = +2	Setup clearance for pre-positioning in the tool axis
12	FN 0: Q12 = +350	Feed rate for milling
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+7.5	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
21	CALL LBL 10	Call machining operation
22	L Z+100 R0 F MAX M2	Retract tool, end of program

23	`LBL 10`	Subprogram 10: Machining operation
24	`FN 1: Q23 = +Q11 + +Q6`	Calculate Z coordinate for pre-positioning
25	`FN 0: Q24 = +Q4`	Copy starting angle in space (Z/X plane)
26	`FN 1: Q26 = +Q6 + +Q108`	Compensate sphere radius for pre-positioning
27	`FN 0: Q28 = +Q8`	Copy rotational position in the plane
28	`FN 1: Q16 = +Q6 + -Q10`	Account for allowance in the sphere radius
29	`CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center of sphere
30	`CYCL DEF 7.1 X+Q1`	
31	`CYCL DEF 7.2 Y+Q2`	
32	`CYCL DEF 7.3 Z-Q16`	
33	`CYCL DEF 10.0 ROTATION`	Account for starting angle of rotational position in the plane
34	`CYCL DEF 10.1 ROT+Q8`	
35	`CC X+0 Y+0`	Set pole in the X/Y plane for pre-positioning
36	`LP PR+Q26 PA+Q8 R0 FQ12`	Pre-position in the plane
37	`LBL 1`	Pre-position in the tool axis
38	`CC Z+0 X+Q108`	Set pole in the Z/X plane, offset by the tool radius
39	`L Y+0 Z+0 FQ12`	Move to working depth
40	`LBL 2`	
41	`LP PR+Q6 PA+Q24 R0 FQ12`	Move upward in an approximated "arc"
42	`FN 2: Q24 = +Q24 - +Q14`	Update solid angle
43	`FN 11: IF +Q24 GT +Q5 GOTO LBL 2`	Inquire whether an arc is finished. If not finished, return to LBL 2.
44	`LP PR+Q6 PA+Q5`	Move to the end angle in space
45	`L Z+Q23 R0 F1000`	Retract in the tool axis
46	`L X+Q26 R0 F MAX`	Pre-position for next arc
47	`FN 1: Q28 = +Q28 + +Q18`	Update rotational position in the plane
48	`FN 0: Q24 = +Q4`	Reset solid angle
49	`CYCL DEF 10.0 ROTATION`	Activate new rotational position
50	`CYCL DEF 10.1 ROT+Q28`	
51	`FN 12: IF +Q28 LT +Q9 GOTO LBL 1`	
52	`FN 9: IF +Q28 EQU +Q9 GOTO LBL 1`	Unfinished? If not finished, return to LBL 1
53	`CYCL DEF 10.0 ROTATION`	Reset rotation
54	`CYCL DEF 10.1 ROT+0`	
55	`CYCL DEF 7.0 DATUM SHIFT`	Reset datum shift
56	`CYCL DEF 7.1 X+0`	
57	`CYCL DEF 7.2 Y+0`	
58	`CYCL DEF 7.3 Z+0`	
59	`LBL 0`	End of subprogram
60	`END PGM BALL MM`	

10

Programming:

Q Parameters

10.1 Principle and Overview

You can program an entire family of parts on the TNC with a single part program. You do this by entering variables called *Q parameters* instead of fixed numerical values.

Q parameters can represent information such as:

- Coordinate values
- Feed rates
- RPM
- Cycle data

Q parameters also enable you to program contours that are defined through mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by the letter Q and a number between 0 and 299. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, locally effective only within the program (depending on MP7251)	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, Q200 to Q299 globally effective for all programs that are stored in the TNC memory	

Programming notes
You can mix Q parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between –99 999.9999 and +99 999.9999.

☞ Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius. For further information, see section "10.9 Preassigned Q Parameters."

Calling Q parameter functions
When you are writing a part program, press the "Q" key (below the –/+ key in the keypad for numerical input and axis selection).

The TNC then displays the following soft keys:

Function group	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITH-METIC
Trigonometric functions	TRIGO-NOMETRY
If/then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Enter formula directly from keyboard	FORMULA

10.2 Part Families — Q Parameters in Place of Numerical Values

The Q parameter function FN0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Resulting NC blocks

15 FN0: Q10 = 25	ASSIGN:
...	Q10 contains the value 25
25 L X +Q10	Means L X +25

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.
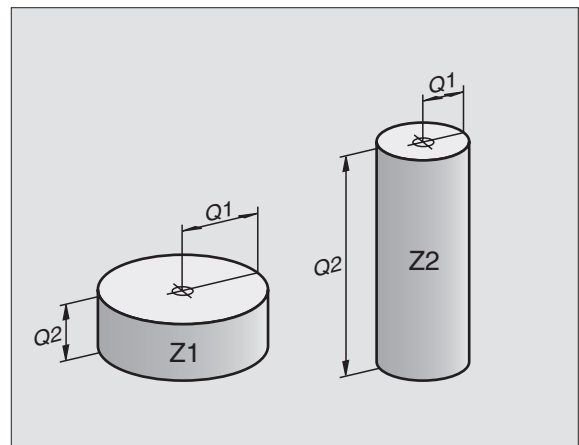
Example
Cylinder with Q parameters

Cylinder radius	R	=	Q1
Cylinder height	H	=	Q2
Cylinder Z1			Q1 = +30
			Q2 = +10
Cylinder Z2			Q1 = +10
			Q2 = +50

10.3 Describing Contours Through Mathematical Functions

The Q parameters listed below enable you to program basic mathematical functions in a part program:

▶ To select the Q parameter function, press the Q key (at the right, in the keypad for numerical input). The Q parameter functions are displayed in a soft-key row.

▶ To select the mathematical functions, press the BASIC ARITHME-TIC soft key. The TNC then displays the following soft keys:

Function	Soft key
FN0: ASSIGN Example: FN0: Q5 = +60 Assigns a numerical value.	FN0 X = V
FN1: ADDITION Example: FN1: Q1 = −Q2 + −5 Calculates and assigns the sum of two values.	FN1 X + V
FN2: SUBTRACTION Example: FN2: Q1 = +10 − +5 Calculates and assigns the difference of two values.	FN2 X − V
FN3: MULTIPLICATION Example: FN3: Q2 = +3 ∗ +3 Calculates and assigns the product of two values.	FN3 X ∗ V
FN4: DIVISION Example: FN4: Q4 = +8 DIV +Q2 Calculates and assigns the quotient of two values. **Not permitted:** division by 0	FN4 X ∕ V
FN5: SQUARE ROOT Example: FN5: Q20 = SQRT 4 Calculates and assigns the square root of a number. **Not permitted:** square root of a negative number	FN5 SQRT

"Values" in the above table are entered to the right of the "=" character and can be any of the following:

■ Two numbers

■ Two Q parameters

■ A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Example: Programming fundamental operations

| Q | To select Q parameter functions, press the Q key. |

| BASIC ARITH-METIC | To select fundamental operations, press the BASIC ARITHMETIC soft key. |

| FN0 X = V | To select the Q parameter function ASSIGN, press the FN0 X = Y soft key. |

PARAMETER NUMBER FOR RESULT?

| 5 ENT | Enter a parameter number, for example 5. |

FIRST VALUE / PARAMETER?

| 1 0 ENT | Assign a value to Q5, for example 10. |

| Q | To select the Q parameter functions, press the Q key. |

| BASIC ARITH-METIC | To select fundamental operations, press the BASIC ARITHMETIC soft key. |

| FN3 X ∗ V | To select the Q parameter function MULTIPLICA-TION, press the FN3 X ∗ Y soft key. |

PARAMETER NUMBER FOR RESULT?

| 1 2 ENT | Enter a Q parameter number, for example 12. |

FIRST VALUE / PARAMETER

| Q 5 ENT | Enter Q5 for the first value. |

SECOND VALUE / PARAMETER

| 7 ENT | Enter 7 for the second value. |

The TNC displays the following program blocks:

```
16 FN0: Q5 = +10
17 FN3: Q12 = +Q5 * +7
```

10.4 Trigonometric Functions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a right triangle, the trigonometric functions of the angle α are defined by the following equations:

Sine: $\sin \alpha = a / c$

Cosine: $\cos \alpha = b / c$

Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

■ c is the side opposite the right angle

■ a is the side opposite the angle α

■ b is the third side.

The TNC can find the angle from the tangent:

$\alpha = \arctan \alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

a = 10 mm

b = 10 mm

$\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions
Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table at right.

Programming method: Trigonometric functions are programmed similar to fundamental operations, see page 223 "Example: Programming fundamental operations."

Function	Soft key
FN6: SINE Example: FN6: Q20 = SIN–Q5 Calculate the sine of an angle in degrees (°) and assign it to a parameter.	FN6 SIN(X)
FN7: COSINE Example: FN7: Q21 = COS–Q5 Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	FN7 COS(X)
FN8: ROOT-SUM OF SQUARES Example: FN8: Q10 = +5 LEN +4 Take the square root of the sum of two squares and assign it to a parameter.	FN8 X LEN V
FN13: ANGLE Example: FN13: Q20 = +10 ANG–Q1 Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.	FN13 X ANG V

10.5 If-Then Decisions with Q Parameters

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see section 9 "Subprograms and Program Section Repeats"). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter PGM CALL after the block with the target label.

Unconditional jumps
An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN9: IF+10 EQU+10 GOTO LBL1

Programming If-Then decisions
Press the JUMP soft key to call the if-then conditions. The TNC then displays the following soft keys:

Function	Soft key
FN9: IF EQUAL, JUMP Example: FN9: IF +Q1 EQU +Q3 GOTO LBL 5 If the two values or parameters are equal, jump to the given label.	FN9 IF X EQ Y GOTO
FN10: IF NOT EQUAL, JUMP Example: FN10: IF +10 NE –Q5 GOTO LBL 10 If the two values or parameters are not equal, jump to the given label.	FN10 IF X NE Y GOTO
FN11: IF GREATERTHAN, JUMP Example: FN11: IF+Q1 GT+10 GOTO LBL 5 If the first value or parameter is greater than the second value or parameter, jump to the given label.	FN11 IF X GT Y GOTO
FN12: IF LESS THAN, JUMP Example: FN12: IF+Q5 LT+0 GOTO LBL 1 If the first value or parameter is less than the second value or parameter, jump to the given label.	FN12 IF X LT Y GOTO

Abbreviations used:

IF If

EQU Equals

NE Not equal

GT Greater than

LT Less than

GOTO Go to

10.6 Checking and Changing Q Parameters

During a program run or test run, you can check or change Q parameters if necessary.

▶ If you are in a program run, interrupt it (for example by pressing the machine STOP button and the INTERNAL STOP soft key). If you are doing a test run, interrupt it.

Q

▶ To call the Q parameter functions, press the Q key.

▶ Enter the Q parameter number and press the ENT key. The TNC displays the current value of the Q parameter in the dialog line.

▶ If you wish to change the value, enter a new value, confirm it with the ENT key and conclude your entry with the END key.

To leave the value unchanged, terminate the dialog with the END key.

```
MANUAL      TEST RUN
OPERATION    Q201 =  -50
0   BEGIN PGM 1 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-40
2   BLK FORM 0.2 X+100 Y+100 Z+0
*   - BOHRPLATTE ID-NR 257943KL1
4   TOOL CALL 1 Z S4500
5   L Z+100 R0 F MAX
6   CYCL DEF 203 UNIVERSL DRILLNG
    Q200=2     ;SET-UP CLEARANCE
    Q201=-50   ;DEPTH
    Q206=250   ;FEED RATE FOR PLNGNG
    Q202=0     ;PECKING DEPTH
    Q210=0     ;DWELL TIME AT TOP
    Q203=+0    ;SURFACE COORDINATE
    Q204=100   ;2ND SET-UP CLEARANCE
    Q212=0     ;DECREMENT

                                    END
```

10.7 Additional Functions

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key
FN14:ERROR Display error messages	FN14 ERROR=
FN15:PRINT Unformatted output of texts or Q parameter values	FN15 PRINT
FN16:F-PRINT Formatted output of texts or Q parameter values	FN16 F-PRINT
FN18:SYS-DATUM READ Read system data	FN18 SYS-DATUM READ
FN19:PLC Transfer values to the PLC	FN19 PLC=

FN14: ERROR
Displaying error messages

With the function FN14: ERROR you can call messages under program control. The messages were preprogrammed by the machine tool builder or by HEIDENHAIN. If the TNC encounters a block with FN 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers and the associated texts are listed in the table at right.

Resulting NC block
The TNC is to display the text stored under error number 254:

```
180 FN 14:ERROR = 254
```

Range of error numbers	Standard dialog text
0 ... 299	FN 14: ERROR CODE 0 299
300 ... 999	No standard dialog text prepared
1000 ... 1099	Internal error messages (see table at right)

Error code and text	
1000	SPINDLE MUST BE TURNING
1001	TOOL AXIS IS MISSING
1002	SLOT WIDTH TOO LARGE
1003	TOOL RADIUS TOO LARGE
1004	RANGE EXCEEDED
1005	START POSITION INCORRECT
1006	ROTATION NOT PERMITTED
1007	SCALING FACTOR NOT PERMITTED
1008	MIRRORING NOT PERMITTED
1009	DATUM SHIFT NOT PERMITTED
1010	FEED RATE IS MISSING
1011	ENTRY VALUE INCORRECT
1012	WRONG SIGN PROGRAMMED
1013	ENTERED ANGLE NOT PERMITTED
1014	TOUCH POINT INACCESSIBLE
1015	TOO MANY POINTS
1016	CONTRADICTORY ENTRY
1017	CYCL INCOMPLETE
1018	PLANE WRONGLY DEFINED
1019	WRONG AXIS PROGRAMMED
1020	WRONG RPM
1021	RADIUS COMP. UNDEFINED
1022	ROUNDING-OFF UNDEFINED
1023	ROUNDING RADIUS TOO LARGE
1024	PROGRAM START UNDEFINED
1025	EXCESSIVE SUBPROGRAMMING
1026	ANGLE REFERENCE MISSING

FN15:PRINT
Unformatted output of texts or Q parameter values

👉 Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see section 14 "MOD Functions, Setting the Date Interface").

The function FN15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode).

To output dialog texts and error messages with FN15: PRINT "numerical value"
Numerical values from 0 to 99:   Dialog texts for OEM cycles

Numerical values exceeding 100: PLC error messages

Example: Output of dialog text 20

```
67 FN15:PRINT 20
```

To output dialog texts and error messages with FN15: PRINT "Q parameter"
Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously. The TNC separates them with slashes.

Example: Output of dialog text 1 and numerical value for Q1

```
70 FN15:PRINT 1/Q1
```

FN16:F-PRINT
Formatted output of texts or Q parameter values

👉 Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see Chapter 14 "MOD Functions, Setting the Date Interface").

The function FN16: F-PRINT transfers Q parameter values and texts in a selectable format through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN16RUN.A (output in program run mode) or in the file %FN16SIM.A (output in test run mode).

To output the formatted texts and Q parameter values, create a text file with the TNC's text editor, and define the output format and Q parameters in this file.

```
MANUAL          PROGRAMMING AND EDITING
OPERATION

RS232 INTERFACE        RS422 INTERFACE

MODE OF OP.: LSV-2    MODE OF OP.: LSV-2
BAUD RATE             BAUD RATE
FE   :    9600        FE   :    9600
EXT1 :    9600        EXT1 :    9600
EXT2 :    9600        EXT2 :    9600
LSV-2:    9600        LSV-2:    9600

ASSIGN:

PRINT      :   TNC:\SCREENS\NEUEBA
PRINT-TEST :
```

| 0-m | RS 232 RS 422 SETUP | USER PARAMETER | HELP | | | | END |

Example of a text file to define the output format:

"TEST RECORD BUCKET WHEEL GRAVITY CENTER";

"_____";

"NO. OF MEASURED VALUES : = 1";

"***";

"X1 = %4.3LF", Q31;

"Y1 = %4.3LF", Q32;

"Z1 = %2I", Q33;

"***";

When you create a text file, use the following formatting functions:

Special character	Function
"............"	Define output format for texts and variables between the quotation marks
%5.4LF	Define format for Q parameters (long, floating): 5 places before and 4 places behind the decimal point
%2I	Define format for Q parameters (integer): Integer with 5 places (in this example: with 2 places)
,	Separation character between output format and parameter
;	End of block character

To activate output, program FN16: F-PRINT in the part program:

`96  FN16:F-PRINT  TNC:\MASK\MASK1.A`

The TNC then transfers the associated file %FN16SIM.A:

```
CALIBRAT.   CHART   IMPELLER   CENTER   GRAVITY

_____

NO.  OF  MEASURED  VALUES  :  =  1
*****************************************
X1  =  149.360
Y1  =  25.509
Z1  =  37
*****************************************
```

FN18:SYS-DATUM READ
Reading system data

With the function FN18: SYS-DATUM READ you can read system data and store them in Q parameters. You select the system data through a group number (ID number), and additionally through a number and an index.

Group name, ID No.	Number	System data
Program information, 10	1	MM/inch condition
	2	Overlap factor for pocket milling
	3	Number of active fixed cycle
Machine status, 20	1	Active tool number
	2	Prepared tool number
	3	Active tool axis
	4	Programmed spindle rpm
	5	Active spindle status
	8	Coolant status
	9	Active feed rate
Data from the tool table, 50	1	Tool length
	2	Tool radius
	3	Tool radius R2
	4	Oversize for tool length DL
	5	Oversize for tool radius DR
	6	Oversize for tool radius DR2
	7	Tool inhibited (0 or 1)
	8	Number of replacement tool
	9	Maximum tool age TIME1
	10	Maximum tool age TIME2
	11	Current tool age CUR. TIME
	12	PLC status
	13	Maximum tooth length LCUTS
	14	Maximum plunge angle ANGLE
	15	TT: Number of teeth CUT
	16	TT: Wear tolerance for length LTOL
	17	TT: Wear tolerance for radius RTOL
	18	TT: Rotational direction DIRECT (3 or 4)
	19	TT: Offset for radius R-OFFS
	20	TT: Offset for length L-OFFS
	21	TT: Breakage tolerance in length LBREAK
	22	TT: Breakage tolerance in radius RBREAK

Group name	Group No.	Number	Index	System data
Active transformations	210	1	—	Basic rotation in MANUAL OPERATION mode
		2	—	Programmed rotation with Cycle 10
		3	—	Active mirror axis
				0: mirroring not active
				+1: X axis mirrored
				+2: Y axis mirrored
				+4: Z axis mirrored
				+8: IV axis mirrored
				+16: V axis mirrored
				Combinations = sum of individual axes
		4	1	Active scaling factor in X axis
		4	2	Active scaling factor in Y axis
		4	3	Active scaling factor in Z axis
		4	4	Active scaling factor in IV axis
		4	5	Active scaling factor in V axis
		5	1	3D ROT A axis
		5	2	3D ROT B axis
		5	3	3D ROT C axis
Data from the last TOOL CALL block	60	1		Tool number
		2		Tool axis
		3		Spindle rpm
		4		Oversize for tool length DL
		5		Oversize for tool radius DR

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
55  FN18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN19:PLC
Transferring values to the PLC

The function FN19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
56  FN19:PLC=+10/+Q3
```

10.8 Entering Formulas Directly

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Entering formulas
Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
Addition Example: Q10 = Q1 + Q5	+
Subtraction Example: Q25 = Q7 – Q108	–
Multiplication Example: Q12 = 5 ∗ Q5	∗
Division Example: Q25 = Q1 / Q2	/
Open parentheses Example: Q12 = Q1 ∗ (Q2 + Q3)	(
Close parentheses Example: Q12 = Q1 ∗ (Q2 + Q3)	)
Square Example: Q15 = SQ 5	SQ
Square root Example: Q22 = SQRT 25	SQRT
Sine of an angle Example: Q44 = SIN 45	SIN
Cosine of an angle Example: Q45 = COS 45	COS
Tangent of an angle Example: Q46 = TAN 45	TAN

Mathematical function	Soft key
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: Q10 = ASIN 0.75	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: Q11 = ACOS Q40	ACOS
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q50	ATAN
Powers Example: Q15 = 3^3	^
π 3.14159	PI
Natural logarithm (LN) of a number Base 2.7183 Example: Q15 = LN Q11	LN
Logarithm of a number, base 10 Example: Q33 = LOG Q22	LOG
Exponential function, 2.7183^n Example: Q1 = EXP Q12	EXP
Negate (multiplication by -1) Example: Q2 = NEG Q1	NEG
Drop places after the decimal point (form an integer) Example: Q3 = INT Q42	INT
Absolute value Example: Q4 = ABS Q22	ABS
Drop places before the decimal point (form a fraction) Example: Q5 = FRAC Q23	FRAC

Rules for formulas

Mathematical formulas are programmed according to the following rules:

■ **Higher-level operations are performed first** (multiplication and division before addition and subtraction)

`12 Q1 = 5 * 3 + 2 * 10 = 35`

1st step: $5 * 3 = 15$
2nd step: $2 * 10 = 20$
3rd step: $15 + 20 = 35$

`13 Q2 = SQ 10 − 3^3 = 73`

1st step: $10^2 = 100$
2nd step: $3^3 = 27$
3rd step: $100 − 27 = 73$

■ **Distributive law**
for calculating with parentheses

$a * (b + c) = a * b + a * c$

Programming example

Calculate an angle with arc tangent as opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q FORMULA		To select the formula entering function, press the Q key and FORMULA soft key.

PARAMETER NUMBER FOR RESULT?

25 ENT		Enter the parameter number.

▷ TAN		Shift the soft-key row and select the arc tangent function.

◁ (		Shift the soft-key row and open parentheses.

Q 12		Enter Q parameter number 12.

/		Select division.

Q 13		Enter Q parameter number 13.

) END		Close parentheses and conclude formula entry.

Resulting NC block

`37  Q25  =  ATAN  (Q12/Q13)`

10.9 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107
The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Tool radius: Q108
The current value of the tool radius is assigned to Q108.

Tool axis: Q109
The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
Z axis	Q109 = 2
Y axis	Q109 = 1
X axis	Q109 = 0

Spindle status: Q110
The value of Q110 depends on which M function was last programmed for the spindle:

M function	Parameter value
No spindle status defined	Q110 = –1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112
The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with PGM CALL) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118
Vth axis	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 110

Actual-nominal deviation	Parameter
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC

Coordinates	Parameter
A axis	Q120
B axis	Q121
C axis	Q122

Example: Ellipse

Process

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating steps you define for the lines, the smoother the curve becomes.

- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
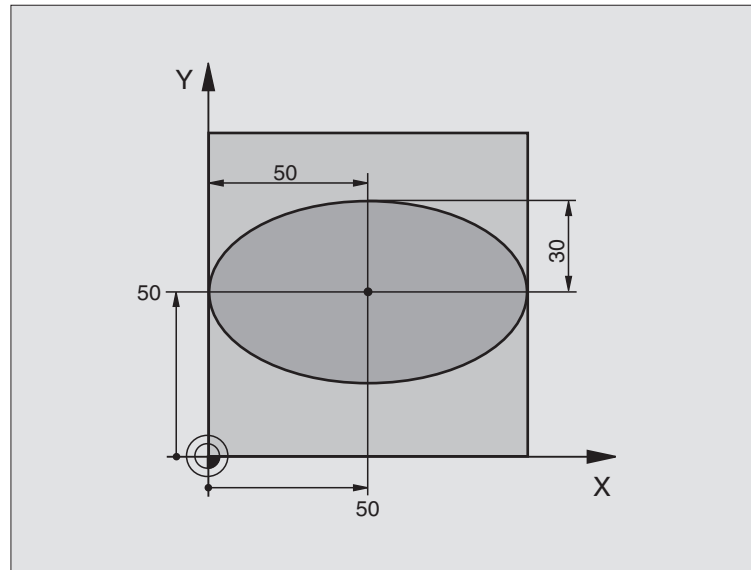
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle

- The tool radius is not taken into account.

0	BEGIN PGM ELLIPSE MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q3 = +50	Semiaxis in X
4	FN 0: Q4 = +30	Semiaxis in Y
5	FN 0: Q5 = +0	Starting angle in the plane
6	FN 0: Q6 = +360	End angle in the plane
7	FN 0: Q7 = +40	Number of calculating steps
8	FN 0: Q8 = +0	Rotational position of the ellipse
9	FN 0: Q9 = +5	Milling depth
10	FN 0: Q10 = +100	Pecking depth
11	FN 0: Q11 = +350	Feed rate for milling
12	FN 0: Q12 = +2	Setup clearance for pre-positioning
13	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+2.5	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	L Z+100 R0 F MAX M2	Retract tool, end of program

20	`LBL 10`	Subprogram 10: Machining operation
21	`CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center of ellipse
22	`CYCL DEF 7.1 X+Q1`	
23	`CYCL DEF 7.2 Y+Q2`	
24	`CYCL DEF 10.0 ROTATION`	Account for rotational position in the plane
25	`CYCL DEF 10.1 ROT+Q8`	
26	`Q35 = (Q6 - Q5) / Q7`	Calculate angle increment
27	`Q36 = Q5`	Copy starting angle
28	`Q37 = 0`	Set counter
29	`Q21 = Q3 * COS Q36`	Calculate X coordinate for starting point
30	`Q22 = Q4 * SIN Q36`	Calculate Y coordinate for starting point
31	`L X+Q21 Y+Q22 R0 F MAX M3`	Move to starting point in the plane
32	`L Z+Q12 R0 F MAX`	Pre-position in tool axis to setup clearance
33	`L Z-Q9 R0 FQ10`	Move to working depth
34	`LBL 1`	
35	`Q36 = Q36 + Q35`	Update the angle
36	`Q37 = Q37 + 1`	Update the counter
37	`Q21 = Q3 * COS Q36`	Calculate the current X coordinate
38	`Q22 = Q4 * SIN Q36`	Calculate the current Y coordinate
39	`L X+Q21 Y+Q22 R0 FQ11`	Move to next point
40	`FN 12: IF +Q37 LT +Q7 GOTO LBL 1`	Unfinished? If not finished return to LBL 1
41	`CYCL DEF 10.0 ROTATION`	Reset rotation
42	`CYCL DEF 10.1 ROT+0`	
43	`CYCL DEF 7.0 DATUM SHIFT`	Reset datum shift
44	`CYCL DEF 7.1 X+0`	
45	`CYCL DEF 7.2 Y+0`	
46	`L Z+Q12 R0 F MAX`	Move to setup clearance
47	`LBL 0`	End of subprogram
48	`END PGM ELLIPSE MM`	

Example: Concave cylinder machined with spherical cutter

Process
- This program requires a spherical cutter.

- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.

- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).

- The machining direction can be altered by changing the entries for the starting and end angles in space:

  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle

- The tool radius is compensated automatically.

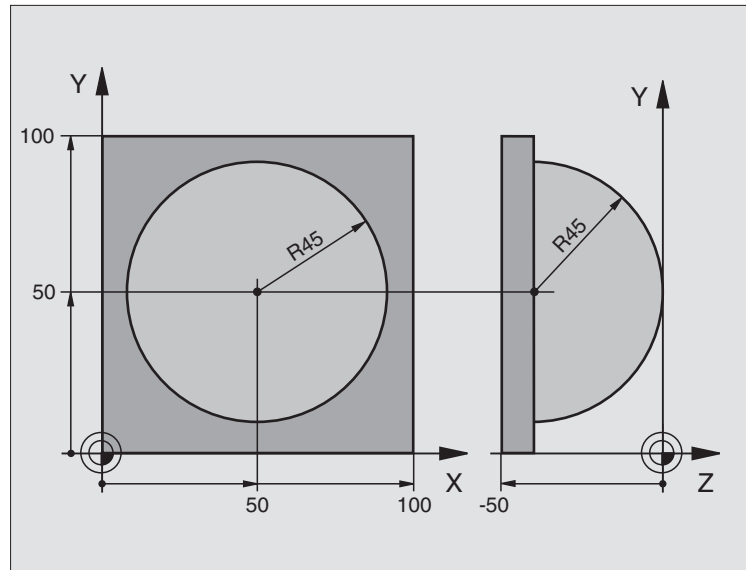| 0	BEGIN PGM CYLIN MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +0	Center in Y axis
3	FN 0: Q3 = +0	Center in Z axis
4	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
5	FN 0: Q5 = +270	End angle in space (Z/X plane)
6	FN 0: Q6 = +40	Radius of the cylinder
7	FN 0: Q7 = +100	Length of the cylinder
8	FN 0: Q8 = +0	Rotational position in the X/Y plane
9	FN 0: Q10 = +5	Allowance for cylinder radius
10	FN 0: Q11 = +250	Feed rate for pecking
11	FN 0: Q12 = +400	Feed rate for milling
12	FN 0: Q13 = +90	Number of cuts
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+3	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	CALL LBL 10	Call machining operation
21	L Z+100 R0 F MAX M2	Retract tool, end of program

22	LBL 10	Subprogram 10: Machining operation
23	Q16 = Q6 - Q10 - Q108	Account for allowance and tool, based on the cylinder radius
24	FN 0: Q20 = +1	Set counter
25	FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
26	Q25 = (Q5 - Q4) / Q13	Calculate angle increment
27	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of cylinder (X axis)
28	CYCL DEF 7.1 X+Q1	
29	CYCL DEF 7.2 Y+Q2	
30	CYCL DEF 7.3 Z-Q3	
31	CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
32	CYCL DEF 10.1 ROT+Q8	
33	L X+0 Y+0 R0 F MAX	Pre-position in the plane to the cylinder center
34	L Z+5 R0 F1000 M3	Pre-position in the tool axis
35	CC Z+0 X+0	Set pole in the Z/X plane
36	LP PR+Q16 PA+Q24 FQ11	Move to starting position on cylinder, plunge-cutting obliquely into the material
37	LBL 1	
38	L Y+Q7 R0 FQ11	Longitudinal cut in Y+ direction
39	FN 1: Q20 = +Q20 + +1	Update counter
40	FN 1: Q24 = +Q24 + +Q25	Update solid angle
41	FN 11: IF +Q20 GT +Q13 GOTO LBL 99	Finished? If finished, jump to end.
42	LP PR+Q16 PA+Q24 FQ12	Move in an approximated "arc" for the next longitudinal cut
43	L Y+0 R0 FQ11	Longitudinal cut in Y– direction
44	FN 1: Q24 = +Q24 + +Q25	Update counter
45	FN 1: Q20 = +Q20 + +1	Update solid angle
46	FN 12: IF +Q20 LT +Q13 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
47	LBL 99	
48	CYCL DEF 10.0 ROTATION	Reset rotation
49	CYCL DEF 10.1 ROT+0	
50	CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
51	CYCL DEF 7.1 X+0	
52	CYCL DEF 7.2 Y+0	
53	CYCL DEF 7.3 Z+0	
54	LBL 0	End of subprogram
55	END PGM CYLIN MM	

Example: Convex sphere machined with end mill

Process

■ This program requires an end mill.

■ The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.

■ You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).

■ The tool moves upward in three-dimensional cuts.

■ The tool radius is compensated automatically.

0	BEGIN PGM BALL MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
4	FN 0: Q5 = +0	End angle in space (Z/X plane)
5	FN 0: Q14 = +5	Angle increment in space
6	FN 0: Q6 = +45	Radius of the sphere
7	FN 0: Q8 = +0	Starting angle of rotational position in the X/Y plane
8	FN 0: Q9 = +360	End angle of rotational position in the X/Y plane
9	FN 0: Q18 = +10	Angle increment in the X/Y plane for roughing
10	FN 0: Q10 = +5	Allowance in sphere radius for roughing
11	FN 0: Q11 = +2	Setup clearance for pre-positioning in the tool axis
12	FN 0: Q12 = +350	Feed rate for milling
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define workpiece blank
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+7.5	Define tool
16	TOOL CALL 1 Z S4000	Call tool
17	L Z+250 R0 F MAX	Retract tool
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
21	CALL LBL 10	Call machining operation
22	L Z+100 R0 F MAX M2	Retract tool, end of program

23	LBL 10	Subprogram 10: Machining operation
24	FN 1: Q23 = +Q11 + +Q6	Calculate Z coordinate for pre-positioning
25	FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
26	FN 1: Q26 = +Q6 + +Q108	Compensate sphere radius for pre-positioning
27	FN 0: Q28 = +Q8	Copy rotational position in the plane
28	FN 1: Q16 = +Q6 + -Q10	Account for allowance in the sphere radius
29	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of sphere
30	CYCL DEF 7.1 X+Q1	
31	CYCL DEF 7.2 Y+Q2	
32	CYCL DEF 7.3 Z-Q16	
33	CYCL DEF 10.0 ROTATION	Account for starting angle of rotational position in the plane
34	CYCL DEF 10.1 ROT+Q8	
35	CC X+0 Y+0	Set pole in the X/Y plane for pre-positioning
36	LP PR+Q26 PA+Q8 R0 FQ12	Pre-position in the plane
37	LBL 1	Pre-position in the tool axis
38	CC Z+0 X+Q108	Set pole in the Z/X plane, offset by the tool radius
39	L Y+0 Z+0 FQ12	Move to working depth
40	LBL 2	
41	LP PR+Q6 PA+Q24 R0 FQ12	Move upward in an approximated "arc"
42	FN 2: Q24 = +Q24 - +Q14	Update solid angle
43	FN 11: IF +Q24 GT +Q5 GOTO LBL 2	Inquire whether an arc is finished. If not finished, return to LBL 2.
44	LP PR+Q6 PA+Q5	Move to the end angle in space
45	L Z+Q23 R0 F1000	Retract in the tool axis
46	L X+Q26 R0 F MAX	Pre-position for next arc
47	FN 1: Q28 = +Q28 + +Q18	Update rotational position in the plane
48	FN 0: Q24 = +Q4	Reset solid angle
49	CYCL DEF 10.0 ROTATION	Activate new rotational position
50	CYCL DEF 10.1 ROT+Q28	
51	FN 12: IF +Q28 LT +Q9 GOTO LBL 1	
52	FN 9: IF +Q28 EQU +Q9 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
53	CYCL DEF 10.0 ROTATION	Reset rotation
54	CYCL DEF 10.1 ROT+0	
55	CYCL DEF 7.0 DATUM SHIFT	Reset datum shift
56	CYCL DEF 7.1 X+0	
57	CYCL DEF 7.2 Y+0	
58	CYCL DEF 7.3 Z+0	
59	LBL 0	End of subprogram
60	END PGM BALL MM	

11

**Test Run
and Program Run**

11.1 Graphics

In the program run modes of operation as well as in the TEST RUN mode, the TNC provides the following three display modes that are selectable via soft key:

■ Plan view

■ Projection in three planes

■ 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

■ the current program has no valid blank form definition

■ no program is selected

With machine parameters 7315 to 7317 you can have the TNC display a graphic even if no tool axis is defined or moved.

☞ A graphic simulation is not possible for program sections or programs in which rotary axis movements or a tilted working plane are defined. In this case, the TNC will display an error message.

Overview of display modes

The TNC displays the following soft keys in the program run and test run modes of operation:

Display mode	Soft key
Plan view	■
Projection in 3 planes	
3-D view	

Limitations during program run

A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text ERROR in the graphics window. The machining process is continued, however.

Plan view

▶ Press the soft key for plan view.

▶ Select the number of depth levels (after shifting the soft-key row). You can choose between 16 or 32 shades of depth. The deeper the surface, the darker the shade.

Plan view is the fastest of the three graphic display modes.

Projection in 3 planes

Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (see "Magnifying details").

In addition, you can shift the sectional planes with the corresponding soft keys:

▶ Press the soft key for projection in three planes.

▶ Shift the soft-key row until the TNC displays the following soft keys:

Function	Soft keys	
Shift the vertical sectional plane to the right or left	⊕	⊕
Shift the horizontal sectional plane upwards or downwards	⊞	⊞

The positions of the sectional planes are visible during shifting.

Coordinates of the line of intersection

At the bottom of the graphics window, the TNC displays the coordinates of the line of intersection, referenced to the workpiece datum. Only the coordinates of the working plane are shown. This function is activated with machine parameter 7310.
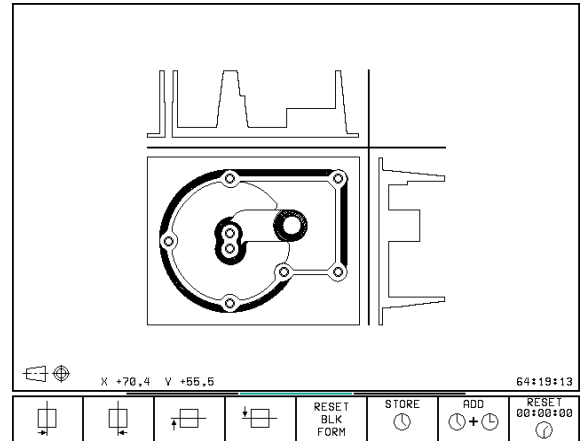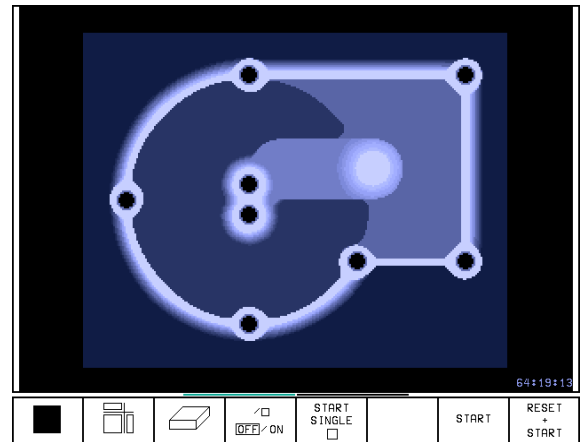
3-D view

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

In the TEST RUN mode of operation you can isolate details for magnification (see "Magnifying details").

▶ Press the soft key for 3-D view.

To rotate the 3-D view
Shift the soft-key row until the following soft keys appear:

Function	Soft keys
Rotate the workpiece in 27° steps about the vertical axis	

Switch the frame overlay display for the workpiece blank on/off:
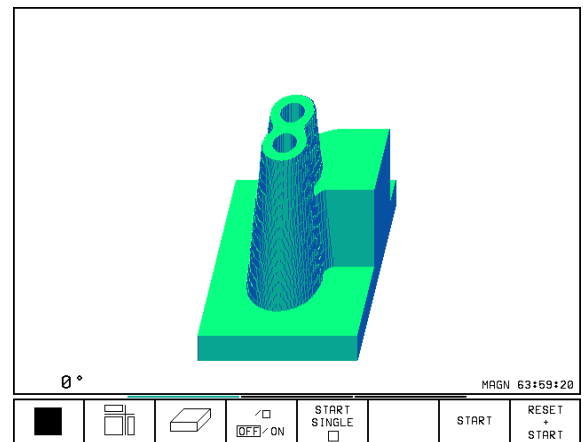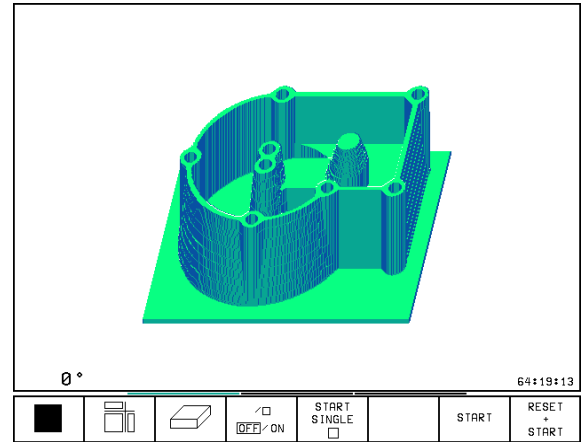
▶ Show the frame overlay with SHOW BLK-FORM

▶ Omit the frame overlay with OMIT BLK-FORM

Magnifying details

You can magnify details in the TEST RUN mode of operation in the following display modes, provided that the graphic simulation is stopped:

■ Projection in three planes

■ 3-D view

A detail magnification is always effective in all three display modes.

Shift the soft-key row in the TEST RUN mode of operation until the following soft keys appear:

Function	Soft keys	
Select the left/right workpiece surface	⬚	⬚
Select the front/back workpiece surface	⬚	⬚
Select the top/bottom workpiece surface	⬚	⬚
Shift the sectional plane to reduce or magnify the blank form	–	+
Select the isolated detail	TRANSFER DETAIL	

To change the detail magnification:
The soft keys are listed in the table above.

▶ Interrupt the graphic simulation, if necessary.

▶ Select the workpiece surface with the corresponding soft key (see table).

▶ To reduce or magnify the blank form, press the minus or plus soft key, respectively.

▶ To select the isolated detail, press the TRANSFER DETAIL soft key.

▶ Restart the test run or program run.

Cursor position during detail magnification
During detail magnification, the TNC displays the coordinates of the axis that is currently being isolated. The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail (MIN point), to the left is the largest (MAX point).

If a graphic display is magnified, this is indicates with MAGN at the lower right of the graphics window.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, enlarge or reduce the workpiece blank.

Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

Function	Soft key
Restore workpiece blank to the detail magnification in which it was last shown	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM	WINDOW BLK FORM

☞ The WINDOW BLK FORM soft key will return the blank form to its original shape or size, even if a detail has been isolated and not yet magnified with TRANSFER DETAIL.

Measuring the machining time

Program run modes of operation
The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

TEST RUN
The timer displays the approximate time which the TNC calculates from the duration of tool movements. The time calculated by the TNC cannot be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

To activate the stopwatch function
Shift the soft-key rows until the TNC displays the following soft keys with the stopwatch functions:

Stopwatch functions	Soft key
Store displayed time	STORE
Display the sum of stored time and displayed time	ADD
Clear displayed time	RESET 00:00:00

☞ The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

```
MANUAL      TEST RUN
OPERATION

0   BEGIN PGM 3DJOINT MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-52
2   BLK FORM 0.2 X+100 Y+100 Z+0
3   TOOL DEF 1 L+0 R+10
4   TOOL CALL 1 Z
5   L Z+20 R0 F MAX M6
6   CYCL DEF 7.0 DATUM SHIFT
7   CYCL DEF 7.1 X-10
8   CALL LBL 1
9   CYCL DEF 7.0 DATUM SHIFT
10  CYCL DEF 7.1 X+0
11  CALL LBL 1
12  CYCL DEF 7.0 DATUM SHIFT
13  CYCL DEF 7.1 X+110
14  CYCL DEF 7.2 Y+100        0°              07:46:04

        SHOW    OMIT    RESET   STORE   ADD    RESET
      BLK-FORM BLK-FORM BLK     ⊕       ⊕+⊕   00:00:00
                        FORM                    ⊘
```

11.2 Functions for Program Display in PROGRAM RUN and TEST RUN

In the program run modes of operation as well as in the TEST RUN mode, the TNC provides the following soft keys for displaying a part program in pages:

Function	Soft key
Go back in the program by one screen	PAGE ⇧
Go forward in the program by one screen	PAGE ⇩
Go to the beginning of the program	BEGIN TEXT
Go to the end of the program	END TEXT

```
PROGRAM RUN, FULL SEQUENCE          PROGRAM TABLE
                                    EDITING

0    BEGIN PGM FK1 MM
1    BLK FORM 0.1 Z X+0 Y+0 Z-20
2    BLK FORM 0.2 X+100 Y+100 Z+0
3    TOOL CALL 1 Z S500
4    L Z+250 R0 F MAX
5    L X-20 Y+30 R0 F MAX
6    L Z-10 R0 F1000 M3
7    APPR CT X+2 Y+30 CCA90 R+5 RL F250
8    FC DR- R18 CLSD+ CCX+20 CCY+30


ACTL.    +X   +250,0000    +Y   +102,3880
         +Z   -114,0914    +C    +30,0000
         +B    +90,0000
T                          F 0        M 5/9
```

PAGE ⇧ | PAGE ⇩ | BEGIN TEXT | END TEXT | RESTORE POS. AT Ⓝ | | ☐ OFF/ON | TOOL TABLE

11.3 Test Run

In the TEST RUN mode of operation you can simulate programs and program sections to prevent errors from occurring during program run. The TNC checks the programs for the following errors:

■ Geometrical incompatibilities

■ Missing data

■ Impossible jumps

■ Violation of the machine's working space

The following functions are also available:

■ Blockwise test run

■ Interrupt test at any block

■ Block skip

■ Graphic simulation

■ Measurement of machining time

■ Additional status display

Running a program test

If the central tool file is active, a tool table must be active (status S) to run a program test. With the MOD function DATUM SET, you can activate work space monitoring for the test run (see Chapter 14 "MOD Functions, Showing the Workpiece in the Working Space").

▶ Select the TEST RUN mode of operation.

▶ Call the file manager with the PGM MGT key and select the file you wish to test, or

▶ Go to the program beginning: Select line "0" with the GOTO key and confirm you entry with the ENT key.

The TNC then displays the following soft keys:

Function	Soft key
Test the entire program	START
Test each program block individually	START SINGLE □
Show the blank form and test the entire program	RESET + START
Interrupt the test run	STOP

Running a program test up to a certain block

With the STOP AT N function the TNC does a test run up to the block with block number N.

▶ Go to the program beginning in the TEST RUN mode of operation.

▶ To do a test run up to a certain block, press the STOP AT N soft key.

STOP AT N ▶ STOP AT N: Enter the block number at which you wish the test to stop.

▶ PROGRAM: Enter the name of the program that contains the block with the selected block number. The TNC displays the name of the selected program. If the test run is to be interrupted in a program that was called with PGM CALL, you must enter this name.

▶ REPETITIONS: If N is located in a program section repeat, enter the number of repeats that you want to run.

▶ To test a program section, press the START soft key. The TNC will test the program up to the entered block.

```
MANUAL          TEST RUN
OPERATION

0   BEGIN PGM 3DJOINT MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-52
2   BLK FORM 0.2 X+100 Y+100 Z+0
3   TOOL DEF 1 L+0 R+10
4   TOOL CALL 1 Z
5   L Z+20 R0 F MAX M6
6   CYCL DEF 7.0 DATUM SHIFT
7   CYCL DEF 7.1 X-10
8   CALL LBL 1
9   CYCL DEF 7.0 DATUM SHIFT
10  CYCL DEF 7.1 X+0

STOP AT:        N = 351
PROGRAM           = 3DJOINT.H
REPETITIONS       = 1
```

| | | | ╱□ OFF╱ON | START SINGLE □ | END | START | RESET + START |

11.4 Program Run

In the PROGRAM RUN, FULL SEQUENCE mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the PROGRAM RUN, SINGLE BLOCK mode of operation you must start each block separately by pressing the machine START button.

The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Block skip
- Editing the tool table TOOL.T
- Checking and changing Q parameters
- Superimposing handwheel positioning
- Functions for graphic simulation
- Additional status display

```
PROGRAM RUN, FULL SEQUENCE          PROGRAM TABLE
                                    EDITING

0   BEGIN PGM FK1 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-20
2   BLK FORM 0.2 X+100 Y+100 Z+0
3   TOOL CALL 1 Z S500
4   L Z+250 R0 F MAX
5   L X-20 Y+30 R0 F MAX
6   L Z-10 R0 F1000 M3
7   APPR CT X+2 Y+30 CCA90 R+5 RL F250
8   FC DR- R18 CLSD+ CCX+20 CCY+30

                         0°              00:00:00

ACTL.   +X  +250,0000   +Y   +102,3880
        +Z  -114,0914   +C    +30,0000
        +B   +90,0000
T                       F 0          M 5/9
```

To run a part program:

Preparation

1 Clamp the workpiece to the machine table.

2 Set the datum.

3 Select the necessary tables and pallet files (status M).

4 Select the part program (status M).

☞ You can adjust the feed rate and spindle speed with the override knobs.

PROGRAM RUN, FULL SEQUENCE
▶ Start the part program with the machine START button.

PROGRAM RUN, SINGLE BLOCK
▶ Start each block of the part program individually with the machine START button.

Interrupting machining

There are several ways to interrupt a program run:

■ Programmed interruptions

■ Machine STOP button

■ Switching to PROGRAM RUN, SINGLE BLOCK

If the TNC registers an error during program run, it automatically interrupts the machining process.

Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts program run at a block containing one of the following entries:

■ STOP (with and without a miscellaneous function)

■ Miscellaneous function M0, M2 or M30

■ Miscellaneous function M6 (determined by the machine tool builder)

To interrupt machining with the machine STOP button:

▶ Press the machine STOP button: The block which the TNC is currently executing is not completed. The asterisk in the status display blinks.

▶ If you do not wish to continue the machining process you can reset the TNC with the INTERNAL STOP soft key. The asterisk in the status display goes out. In this case, the program must be restarted from the program beginning.

To interrupt machining at the end of the current block:

You can interrupt a program that is being run in the PROGRAM RUN, FULL SEQUENCE mode of operation by switching to PROGRAM RUN, SINGLE BLOCK. The TNC interrupts the machining process at the end of the current block.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the MANUAL OPERATION mode.

☞ **Danger of collision**

If you interrupt program run while the working plane is tilted, you can change from a tilted to a non-tilted coordinate system, and vice versa, by pressing the 3D ON/OFF soft key.

The functions of the axis direction buttons, the electronic handwheel and the positioning logic for return to contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3D ROT menu.

Example:
Retracting the spindle after tool breakage

▷ Interrupt machining.

▷ To enable the machine axis direction buttons, press the MANUAL OPERATION soft key.

▷ Move the axes with the machine axis direction buttons.

✎ On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to the operating manual of your machine tool for further information.

Resuming program run after an interruption

☞ If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

■ The data of the last defined tool

■ Active coordinate transformations

■ The coordinates of the circle center that was last defined

```
PROGRAM RUN, FULL SEQUENCE          PROGRAMMING
                                    AND EDITING

7   CYCL DEF 220 POLAR PATTERN
    Q216=+50    ;CENTER IN 1ST AXIS
    Q217=+50    ;CENTER IN 2ND AXIS
    Q244=80     ;PITCH CIRCLE DIA.
    Q245=+0     ;STARTING ANGLE
    Q246=+342   ;STOPPING ANGLE
    Q247=0      ;STEPPING ANGLE
    Q241=20     ;NR OF REPETITIONS
    Q200=2      ;SET-UP CLEARANCE


NOML.     X    +90,0000    Y    +50,0000
*         Z    -0,3875     V    -1,2753
          W    -3,3042
T 1    Z      S 2000     F 0       M 3/9
┌─────────┐                        ┌─────────┐
│ MANUAL  │                        │INTERNAL │
│OPERATION│                        │  STOP   │
└─────────┘                        └─────────┘
```

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION).

Resuming program run with the START button
You can resume program run by pressing the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed

- A programmed interruption

Resuming program run after an error
- If the error message *is not blinking*:

▶ Remove the cause of the error.

▶ To clear the error message from the screen, press the CE key.

▶ Restart the program, or resume program run at the place at which it was interrupted.

- If the error message *is blinking*:

▶ Switch off the TNC and the machine.

▶ Remove the cause of the error.

▶ Start again.

If you cannot correct the error, write down the error message and contact your repair service agency.

Mid-program startup (block scan)

> The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Refer to the operating manual of your machine tool for further information.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

☞ Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files must be selected in a program run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

If you are working with nested programs, you can use machine parameter 7680 to define whether the block scan is to begin at block 0 of the main program or at block 0 of the last interrupted program.

If the working plane is tilted, you can use the 3D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in a non-tilted coordinate system.

▶ To go to the first block of the current program to start a block scan, enter GOTO "0".

▶ To select mid-program startup, press the RESTORE POS. AT N soft key.

RESTORE POS. AT N
▶ START-UP AT N: Enter the block number N at which the block scan should end.

▶ PROGRAM: Enter the name of the program containing block N.

▶ REPETITIONS: If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.

▶ To start the block scan, press the machine START button.

▶ To return to the contour, proceed as described below in "Returning to the contour."

```
PROGRAM RUN, FULL SEQUENCE          TEST RUN

0   BEGIN PGM FK1 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-20
2   BLK FORM 0.2 X+100 Y+100 Z+0
3   TOOL CALL 1 Z S500
4   L Z+250 R0 F MAX

START-UP AT:   N = 35
PROGRAM          = FK1.H
REPETITIONS      = 1


ACTL.   +X   +250,0000   +Y   +102,3880
        +Z   -114,0914   +C    +30,0000
        +B    +90,0000
T                         F 0      M 5/9
PAGE⇧  PAGE⇩  BEGIN  END              ▢        END
              TEXT   TEXT          OFF/ON
```

Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

■ Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.

■ Return to the contour after a block scan with RESTORE POS. AT N, for example after an interruption with INTERNAL STOP.

▶ To select a return to contour, press the RESTORE POSITION soft key.

▶ To move the axes in the sequence that the TNC suggests on the screen, press the POSIT. LOGIC soft key and the machine START key, or

▶ To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.

▶ To resume machining, press the machine START key.

```
PROGRAM RUN, FULL SEQUENCE        PROGRAMMING
                                  AND EDITING

RETURN TO CONTOUR: SEQUENCE OF AXES:
                   X
                   Y
                   Z


-OR ENTER ACCORDING TO SOFT KEY


NOML.     X    +167,0000     Y    +120,0000
          Z     +76,6125     V     -1,2753
          W     -3,3042
T 1   Z        S 2000       F 0        M 3/9
RESTORE  RESTORE  RESTORE              MANUAL   INTERNAL
  X        V        Z                 OPERATION   STOP
```

11.5 Optional Block Skip

In a test run or program run, the TNC can skip over blocks that begin with a slash (/):

▶ To run or test the program with the blocks preceded by a slash, set the soft key to ON.

▶ To run or test the program without the blocks preceded by a slash, set the soft key to OFF.

☞ This function does not work for TOOL DEF blocks.

12

3-D Touch Probes

12.1 Touch Probe Cycles in the MANUAL and ELECTRONIC HAND-WHEEL Operating Modes

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.

If you are carrying out measurements during program run, be sure that the tool data (length, radius, axis) can be used from the calibrated data or from the last TOOL CALL block (selected with MP7411).

If you are working alternately with a triggering and a measuring touch probe, be sure that

- the correct touch probe is selected with MP 6200
- the measuring and the triggering touch probes are never connected to the control at the same time.

The TNC cannot tell which probe is actually in the spindle.

After you press the machine START button, the touch probe approaches the workpiece in a paraxial movement in the selected probe function. The machine tool builder sets the probe feed rate (see figure at right). When the probe contacts the workpiece, it

- transmits a signal to the TNC: the coordinates of the probed position are stored,
- stops moving, and
- returns to its starting position in rapid traverse.

If the stylus is not deflected within a defined distance, the TNC displays an error message (MP 6130 defines the distance for a triggering touch probe, MP6330 for a measuring touch probe).

To select the touch probe functions:
▶ Select the MANUAL OPERATION or ELECTRONIC HAND-WHEEL mode of operation.

TOUCH PROBE   ▶ To choose the touch probe functions, press the TOUCH PROBE soft key. The TNC displays additional soft keys — see table at right.
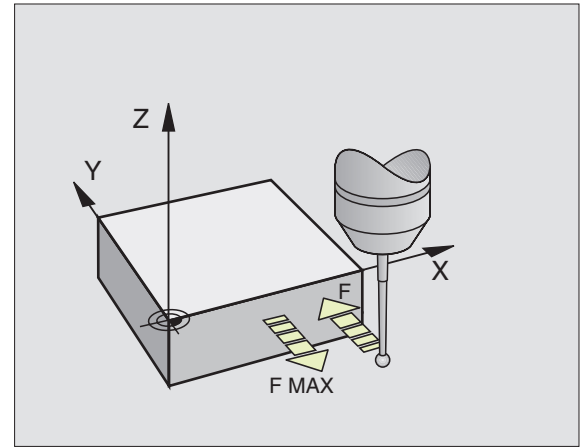
Function	Soft key
Calibrate the effective length	CAL L
Calibrate the effective radius	CAL R 3D CAL
Basic rotation	PROBING ROT
Set the datum	PROBING POS
Set the datum at a corner	PROBING P
Set the datum at a circle center	PROBING CC

Recording the values measured in a probe cycle

🖱 The TNC must be specially prepared by the machine tool builder for use of this function. Refer to the operating manual of your machine tool for further information.

After executing any selected touch probe cycle, the TNC displays the soft key PRINT. If you press this soft key, the TNC will record the current values determined in the active probe cycle. You can then use the PRINT function in the menu for setting the data interface (see "14 MOD Function, Setting the Data Interface") to define whether the TNC is to

■ print the measuring result,

■ store the measuring results on the TNC's hard disk, or

■ store the measuring results on a PC.

If you store the measuring results, the TNC creates the ASCII file %TCHPRNT.A. Unless you define a specific path and interface in the interface configuration menu, the TNC will store the %TCHPRNT file in the main directory TNC:\ .

👉 When you press the PRINT soft key, the %TCHPRNT.A file must not be active in the PROGRAMMING AND EDITING mode of operation. The TNC will otherwise display an error message.

The TNC stores the measured data in the %TCHPRNT.A file only. If you execute several touch probe cycles in succession and want to store the resulting measured data, you must make a backup of the contents stored in %TCHPRNT.A between the individual cycles by copying or renaming the file.

Format and contents of the %TCHPRNT file are preset by the machine tool builder.

Calibrating a triggering touch probe

The touch probe must be calibrated

■ for commissioning

■ when the stylus breaks

■ when the stylus is changed

■ when the probe feed rate is changed

■ in case of irregularities such as those resulting from warming of the machine.

During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known internal radius to the machine table.

```
MANUAL        PROGRAMMING AND EDITING
OPERATION

FILE: %TCHPRNT          LINE: 0     COLUMN: 1   INSERT

■M KALIBRIEREN:
---------------
30-01-1996, 16:00:46

ACHSE TCH PROBE          : Z
TASTERRADIUS 1           : 1.500 MM
TASTERRADIUS 2           : 1.500 MM
DURCHMESSER EINSTELLRING : 50.001 MM
KORREKTURFAKTOR          : X = 1.0000
                         : V = 1.0000
                         : Z = 1.0000
KRAFTVERHAELTNIS         : FX/FZ = 1.0000
                         : FV/FZ = 1.0000

[END]
```

| INSERT OVERWRITE | MOVE WORD >> | MOVE WORD << | PAGE ⇩ | PAGE ⇧ | BEGIN TEXT | END TEXT | FIND |

To calibrate the effective length:

▶ Set the datum in the tool axis such that for the machine tool table Z=0.

CAL L

▶ To select the calibration function for the touch probe length, press the TOUCH PROBE and CAL L soft keys. The TNC then displays a menu window with four input fields.

▶ Enter the TOOL AXIS.

▶ DATUM: Enter the height of the ring gauge.

▶ The menu items EFFECTIVE BALL RADIUS and EFFECTIVE LENGTH do not require input.

▶ Move the touch probe to a position just above the ring gauge.

▶ To change the displayed traverse direction (if necessary), press an arrow key.

▶ To probe the upper surface of the ring gauge, press the machine START button.
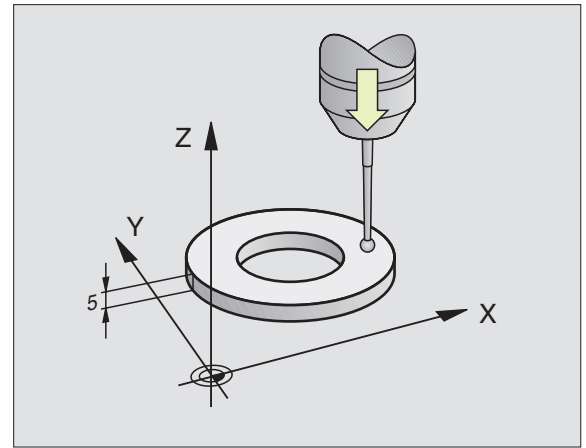
Calibrating the effective radius and compensating center misalignment

After the touch probe is inserted it normally needs to be exactly aligned with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

For this operation, the TNC rotates the 3-D touch probe by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in the machine parameter 6160.

The center misalignment is measured after the effective ball tip radius is calibrated.

▶ In the MANUAL OPERATION mode, position the ball tip in the bore of the ring gauge.

CAL R

▶ To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL R soft key.

▶ Enter the TOOL AXIS. The remaining menu items do not require input.

▶ To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.

▶ If you want to terminate the calibration function at this point, press the END soft key.

180°

▶ If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°.

▶ To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the ball-tip center misalignment.
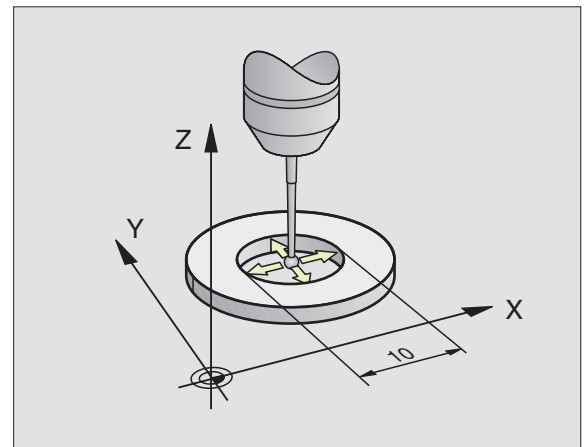
Displaying calibration values

The TNC stores the effective length and radius, and the center misalignment for use when the touch probe is needed again. You can display the values on the screen with the soft keys CAL L and CAL R.

Calibrating a measuring touch probe

☞ If the TNC displays the error message STYLUS AL-READY IN CONTACT, select the 3-D calibration menu and press the RESET 3D soft key.

The measuring touch probe must be calibrated whenever the machine parameters for 3-D touch probes are changed.

The effective length is calibrated in the same way as with triggering touch probes. You must also enter tool radius R2 (corner radius).

In MP6321 you can define whether or not the TNC should probe to find the stylus center.

The 3-D calibration cycle for measuring touch probes enables you to measure a standard ring gauge fully automatically. (The standard ring gauge is available from HEIDENHAIN). Fix the standard ring gauge to the machine table with fixing clamps.

From the data measured during calibration, the TNC calculates the spring rate of the touch probe, the stylus deflection and the stylus center misalignment. At the end of the calibration cycle, the TNC automatically stores these values in the input menu.

▶ In the MANUAL OPERATION mode, position the touch probe to a position approximately in the center of the standard ring gauge and set it to 180°.

▶ To select the 3-D calibration cycle, press the 3D CAL soft key.

▶ Enter the values for STYLUS RADIUS 1 and STYLUS RADIUS 2. Enter the same value for stylus radius 1 and 2 if you are using a stylus with ball tip. Enter different values for stylus radius 1 and 2 if you are using a stylus with a corner radius.

▶ DIAMETER RING GAUGE: The diameter is engraved on the standard ring gauge.

▶ To start the calibration cycle, press the machine START button: The touch probe measures the standard ring gauge in a programmed sequence of steps.

▶ Rotate the touch probe to 0° as soon as the TNC asks you to.

▶ To start the calibration cycle once again to determine center misalignment, press the machine START button. The touch probe again measures the standard ring gauge in a programmed sequence of steps.

```
MANUAL OPERATION                    PROGRAMMING
X+   X-   Y+   Y-                   AND EDITING



TOOL AXIS = Z
RADIUS RING GAUGE = 25
EFFECTIVE PROBE RADIUS = 1,989
EFFECTIVE LENGTH = +0
STYL.TIP CENTER OFFSET X=+0
STYL.TIP CENTER OFFSET Y=+0


ACTL.   +X   +250,0000   +Y   +102,3880
        +Z   -114,0914   +C    +30,0000
        +B    +90,0000
T                           F 0      M 5/9

PRINT                                  END
```

Displaying calibration values

The compensation factors and force ratios are stored in the TNC for later use whenever the measuring touch probe is needed.

You can display the values on the screen by pressing the 3D CAL soft key.

Compensating workpiece misalignment

The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

You set the rotation angle to the desired angle with respect to the reference axis in the working plane (see second figure at right).

☞ Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.

To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.

PROBING ROT ▶ To select the probe function, press the PROBING ROT soft key.

▶ Position the ball tip at a starting position near the first touch point.

▶ Select the probe direction perpendicular to the angle reference axis: Select the axis with the arrow keys.

▶ To probe the workpiece, press the machine START button.

▶ Position the ball tip at a starting position near the second touch point.
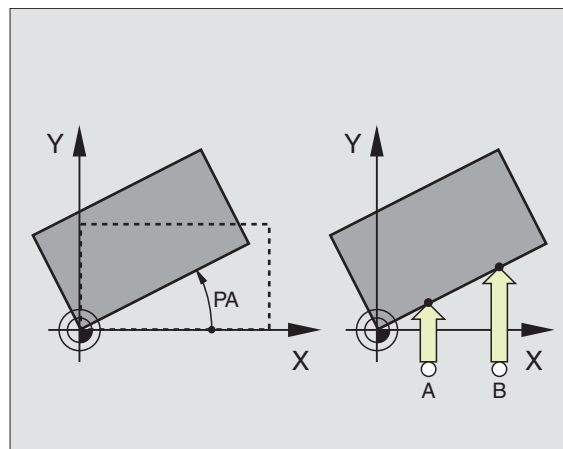
▶ To probe the workpiece, press the machine START button.

The TNC stores a basic rotation in nonvolatile storage. The basic rotation is effective for all subsequent program runs and graphic simulation.

Displaying a basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (STATUS POS.).

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.

```
MANUAL OPERATION                        PROGRAMMING
ROTATE TOUCH PROBE TO 180 DEG           AND EDITING

PROBE TIP RADIUS 1 = 1,5
PROBE TIP RADIUS 2 = 1,5
RING GAUGE DIAMETER = 50,0008
COMPENSATION FACTOR X:1
COMPENSATION FACTOR Y:1
COMPENSATION FACTOR Z:1
SPRING FORCE RATIO FX/FZ:1
SPRING FORCE RATIO FY/FZ:1


ACTL.    +X   +250,0000    +Y   +102,3880
         +Z   -114,0914    +C    +30,0000
         +B    +90,0000
T                          F  0       M 5/9

PRINT                              RESET   END
                                    3D
```



```
MANUAL OPERATION                        PROGRAMMING
X+   X-   Y+   Y-                        AND EDITING




ROTATION ANGLE = +15,235


ACTL.    +X   +250,0000    +Y   +102,3880
         +Z   -114,0914    +C    +30,0000
         +B    +90,0000
T                          F  0       M 5/9

PRINT                                       END
```

To cancel a basic rotation:

▶ To select the probe function, press the PROBING ROT soft key.

▶ Enter a ROTATION ANGLE of zero and confirm with the ENT key.

▶ To terminate the probe function, press the END key.

## 12.2 Setting the Datum with a 3-D Touch Probe

The following functions are available for setting the datum on an aligned workpiece:

■ Datum setting in any axis with PROBING POS

■ Defining a corner as datum with PROBING P

■ Setting the datum at a circle center with PROBING CC

**To set the datum in any axis (see figure to the upper right):**

▶ To select the probe function, press PROBING POS.

▶ Position the touch probe at a starting position near the first touch point.

▶ Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made with the arrow keys.

▶ To probe the workpiece, press the machine START button.

▶ DATUM: Enter the nominal coordinate and confirm you entry with ENT.
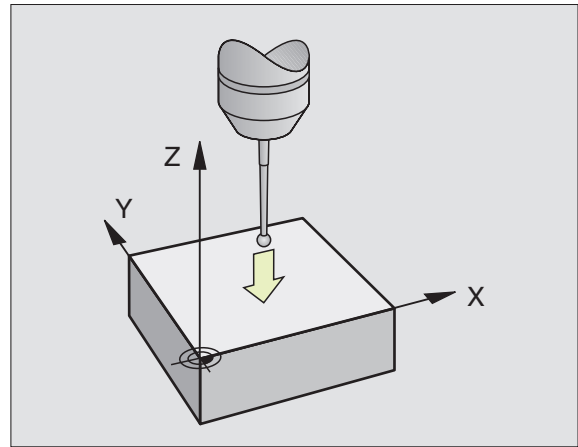
Corner as datum — using points that were already probed for a basic rotation (see figure at right)

▶ To select the probe function, press PROBING P.

▶ TOUCH POINTS OF BASIC ROTATION?: Press ENT to transfer the touch point coordinates to memory.

▶ Position the touch probe at a starting position near the first touch point of the side that was not probed for basic rotation.

▶ Select the probe direction: Select the axis with the arrow keys.

▶ To probe the workpiece, press the machine START button.

▶ Position the touch probe near the second touch point on the same side.

▶ To probe the workpiece, press the machine START button.

▶ DATUM: Enter both datum coordinates into the menu window, and confirm your entry with the ENT key.

▶ To terminate the probe function, press the END key.
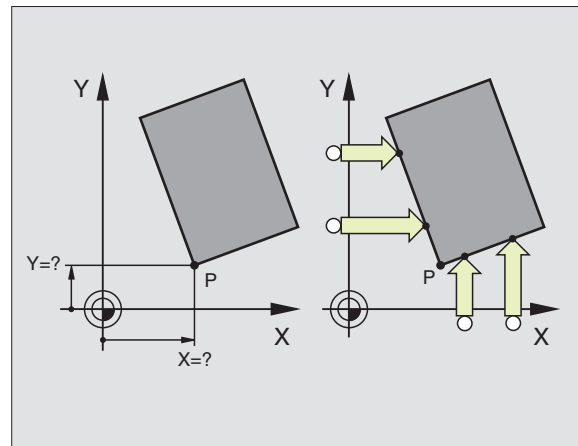
Corner as datum — without using points that were already probed for a basic rotation

▶ To select the probe function, press PROBING P.

▶ TOUCH POINTS OF BASIC ROTATION?: Press NO ENT to ignore the previous touch points. (The dialog question only appears if a basic rotation was made previously.)

▶ Probe both workpiece sides twice.

▶ Enter the coordinates of the datum and confirm your entry with ENT.

▶ To terminate the probe function, press the END key.

Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

Inside circle:

The TNC automatically probes the inside wall in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

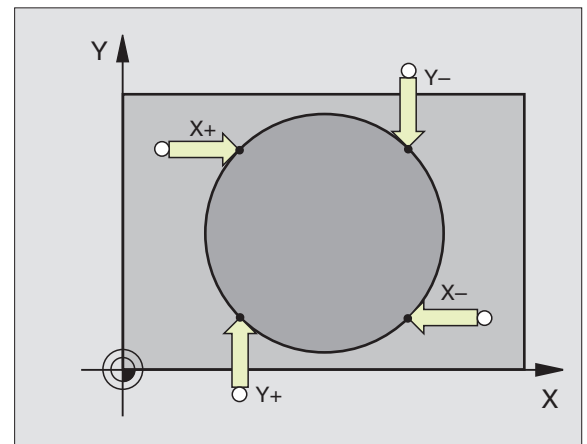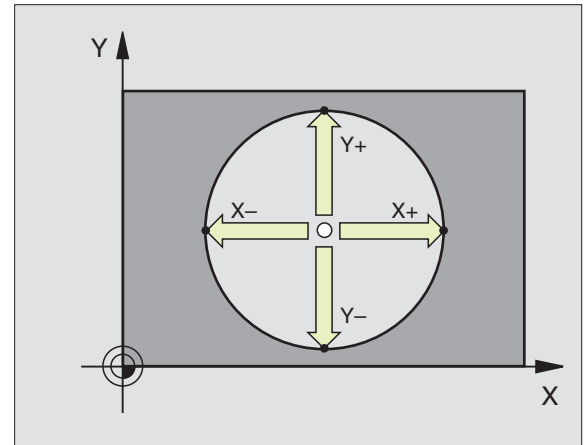▶ Position the touch probe approximately in the center of the circle.

▶ To select the probe function, press PROBING CC.

▶ To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.

▶ If you are probing to find the stylus center (only available on machines with spindle orientation, depending on MP6160), press the 180° soft key and probe another four points on the inside of the circle.

▶ If you are not probing to find the stylus center, press the END key.

▶ DATUM: Enter both circle center coordinates into the menu window, and confirm your entry with ENT.

▶ To terminate the probe function, press the END key.

Outside circle:

▶ Position the touch probe at the starting position for the first touch point outside of the circle.

▶ Select the probing direction: Choose the axis with the arrow keys.

▶ To probe the workpiece, press the machine START button.

▶ Repeat the probing process for the next three points (see figure at lower right).

▶ Enter the datum coordinates and confirm your entry with ENT.

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

Setting datum points over holes

A second soft-key row provides soft keys for using holes to set datums.

The touch probe is used in the same way as in the "circle center as datum — inside corner" function. First pre-position the touch probe in the approximate center of a hole, then press the machine START button to automatically probe four points in the hole.

Move the touch probe to the next hole and have the TNC repeat the probing procedure until all the holes have been probed to set datums.

Function	Soft key
Basic rotation from 2 holes: The TNC measures the angle between the line connecting the centers of two holes and a nominal position (angle reference axis).	PROBING ROT
Basic rotation from 4 holes: The TNC calculates the intersection of the line connecting the first two probed holes with the line connecting the last two probed holes. If a basic rotation was already made from the first two holes, these holes do not need to be probed again.	PROBING P
Circle center from 3 holes: The TNC calculates a circle that intersects the centers of all three holes, and finds the center.	PROBING CC

```
MANUAL OPERATION            PROGRAMMING
                            AND EDITING




ACTL.   +X   +250,0000    +Y   +102,3880
        +Z  -114,0914     +C    +30,0000
        +B   +90,0000
T                        F  0          M 5/9
      PROBING   PROBING PROBING          END
        ROT        P      CC
```

12.3 Measuring Workpieces with a 3-D Touch Probe

With a 3-D touch probe you can determine:

■ position coordinates, and from them,

■ dimensions and angles on the workpiece.

To find the coordinate of a position on an aligned workpiece:

▶ To select the probe function, press the PROBING POS soft key.

▶ Position the touch probe at a starting position near the touch point.

▶ Select the probe direction and axis of the coordinate. Use the arrow keys for selection.

▶ To probe the workpiece, press the machine START button.

The TNC shows the coordinates of the touch point as DATUM.

Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point as described under "Corner as datum." The TNC displays the coordinates of the probed corner as datum.

Measuring workpiece dimensions

▶ To select the probe function, press PROBING POS.

▶ Position the touch probe at a starting position near the first touch point A.

▶ Select the probe direction with the arrow keys.

▶ To probe the workpiece, press the machine START button.

▶ If you will need the current datum later, write down the value that appears in the DATUM display.

▶ DATUM: Enter "0".

▶ To terminate the dialog, press the END key.

▶ Select the probe function again with the PROBING POS soft key.
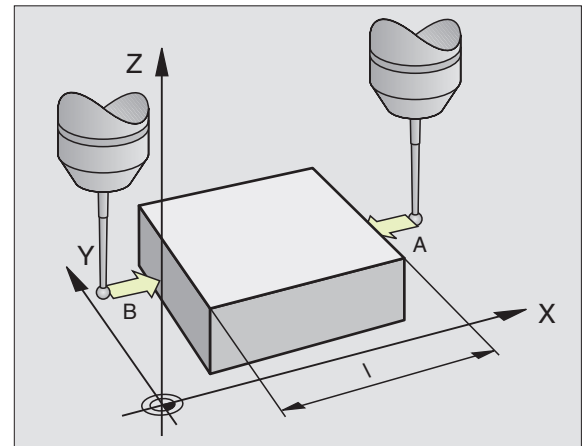
▶ Position the touch probe at a starting position near the second touch point B.

▶ Select the probe direction with the arrow keys: Same axis but opposite direction as for A.

▶ To probe the workpiece, press the machine START button.

The value displayed as DATUM is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

▶ Select the probe function with the PROBING POS soft key.

▶ Probe the first touch point again.

▶ Set the DATUM to the value that you wrote down previously.

▶ To terminate the dialog, press the END key.

Measuring angles
You can also use the touch probe to measure angles in the working plane. You can measure

■ the angle between the angle reference axis and a workpiece side, or

■ the angle between two sides.

The measured angle is displayed as a value of maximum 90°.

To find the angle between the angle reference axis and a side of the workpiece:

▶ Select the probe function with the PROBING ROT soft key.

▶ ROTATION ANGLE: If you will need the current basic rotation later, write down the value that appears under ROTATION ANGLE.

▶ Make a basic rotation with the side of the workpiece (see "Compensating workpiece misalignment").

▶ Press the PROBING ROT soft key to display the angle between the angle reference axis and the side of the workpiece as the ROTATION ANGLE.

▶ Cancel the basic rotation, or restore the previous basic rotation:

▶ This is done by setting the ROTATION ANGLE to the value that you wrote down previously.

To measure the angle between two workpiece sides:

▶ Select the probe function with the PROBING ROT soft key.

▶ ROTATION ANGLE: If you will need the current basic rotation later, write down the value that appears under ROTATION ANGLE.

▶ Make a basic rotation for the first side (see "Compensating workpiece misalignment").

▶ Probe the second side as for a basic rotation, but do *not* set the ROTATION ANGLE to zero!

▶ Press the PROBING ROT soft key to display the angle PA between the two sides as the ROTATION ANGLE.

▶ Cancel the basic rotation, or restore the previous basic rotation by setting the ROTATION ANGLE to the value that you wrote down previously.
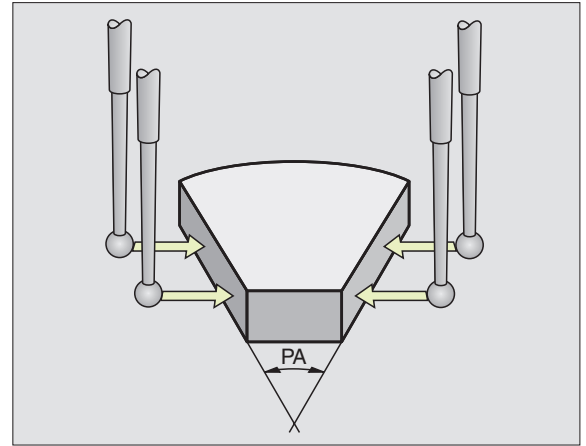
Measuring with the 3-D touch probe during program run

The 3-D touch probe can measure positions on the workpiece while the program is being run — even if the working plane is tilted. Applications:

■ Measuring differences in the height of cast surfaces

■ Tolerance checking during machining

To program the use of a touch probe, press the TOUCH PROBE key in the PROGRAMMING AND EDITING mode of operation. The TNC pre-positions the touch probe to automatically probe the desired position. During probing, the TNC moves the touch probe parallel to the machine axis that was defined in the touch probe cycle. The TNC takes an active basic rotation or rotation only into account for calculating the touch point. The coordinate of the touch point is stored in a Q parameter. The TNC interrupts the probing process if the stylus is not deflected within a certain distance (selectable via MP 6130). Upon contact, the position coordinates of the probe are also stored in the parameters Q115 to Q119. The values in these parameters do not include the stylus length and radius.

☞ Pre-position the touch probe manually to avoid a collision when the programmed pre-positioning point is approached.

Be sure to use the tool data (length, radius, axis) either from the calibrated data or from the last TOOL CALL block. Selection is made with machine parameter MP7411.
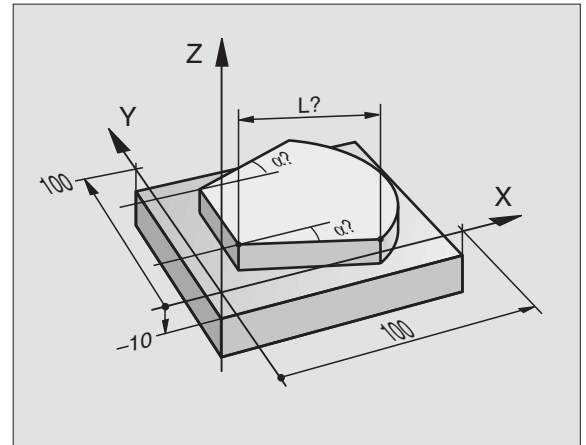
▶ Press the TOUCH PROBE key in the PROGRAMMING AND EDITING mode of operation.

▶ TCH PROBE 0: REF. PLANE: Select the probe function with the ENT key.

▶ PARAMETER NUMBER FOR RESULT: Enter the number of the Q parameter to which you want to assign the coordinate.

▶ PROBING AXIS/PROBING DIRECTION: Enter the probing axis with the axis selection keys and algebraic sign for the probing direction. Confirm you entries with the ENT key.

▶ POSITION VALUE: Use the axis selection keys to enter all coordinates of the nominal pre-positioning point values for the touch probe.

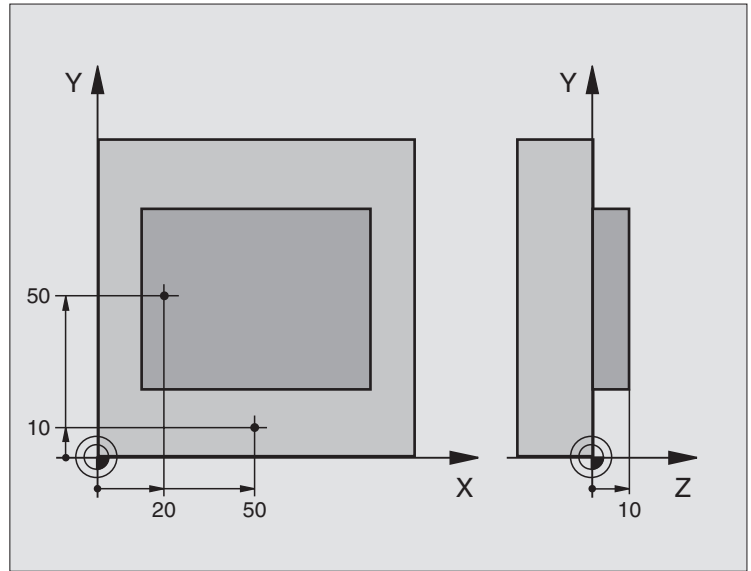▶ To conclude input, press the ENT key.

Resulting NC blocks

```
67 TCH PROBE 0.0 REF. PLANE Q5 X-
```
```
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

Example: Measuring the height of an island on a workpiece

Process

- Assign the program parameters
- Measure the height with the TCH PROBE cycle
- Calculate the height

`0    BEGIN PGM 3DPROBE MM`	
`1   FN 0: Q11 = +20`	1st touch point: X coordinate
`2   FN 0: Q12 = +50`	1st touch point: Y coordinate
`3   FN 0: Q13 = +10`	1st touch point: Z coordinate
`4   FN 0: Q21 = +50`	2nd touch point: X coordinate
`5   FN 0: Q22 = +10`	2nd touch point: Y coordinate
`6   FN 0: Q23 = +0`	2nd touch point: Z coordinate
`7   TOOL CALL 0 Z`	Call touch probe
`8   L Z+250 R0 F MAX`	Retract touch probe
`9   TCH PROBE 0.0 REF. PLANE Q10 Z-`	Measure upper surface of the workpiece
`10  TCH PROBE 0.1 X+Q11 Y+Q12 Z+Q13`	
`11  L X+Q21 Y+Q22 R0 F MAX`	Pre-position for second measurement
`12  TCH PROBE 0.0 REF. PLANE Q20 Z-`	Measure the depth
`13  TCH PROBE 0.1 Z+Q23`	
`14  FN 2: Q1 = +Q20 - +Q10`	Calculate the absolute height of the island
`15  STOP`	Interrupt program run: check Q1
`16  L Z+250 R0 F MAX M2`	Retract in the tool axis, end of program
`17  END PGM 3DPROBE MM`	

13

Digitizing

The digitizing option enables you to reduce a three-dimensional part into discrete digital information by scanning it with a touch probe.

The following components are required for digitizing:

■ Touch probe

■ "Digitizing option" software module in the TNC

■ SUSA evaluation software from HEIDENHAIN for further processing of the data digitized in the MEANDER cycle.

The touch probe systems provide four digitizing cycles:

■ RANGE (cuboid, or table with measuring touch probes)

■ MEANDER

■ CONTOUR LINES

■ LINE

☞ The TNC and the machine tool must be specially prepared by the machine tool builder for the use of a touch probe.

The touch probe must be calibrated before digitizing.

If you are working alternately with a triggering and a measuring touch probe, ensure that

– you have selected the correct touch probe in MP 6200

– the measuring and triggering touch probes are never connected to the TNC at the same time.

The TNC cannot tell which touch probe is actually in the spindle.

Principle

The touch probe scans a 3-D surface point for point in a selectable grid. The scanning speeds for triggering touch probes vary from 200 to 800 mm/min with a probe point interval (PP.INT) of 1 mm. The scanning speeds for measuring touch probes are programmed in the scanning cycle. You can enter up to 3000 mm/min.

The TNC saves the digitized positions directly to the hard disk. The interface function PRINT determines in which directory the data are stored.

If you want to mill the digitized data subsequently using a tool whose radius equals the radius of the ball tip, you can machine the digitized data directly from the file with Cycle 30 (see section 8.7 "Cycles for Multipass Milling").

> Digitizing cycles are programmed only for the main axes X, Y and Z and for the rotary axes A, B and C.
>
> Coordinate transformations or a basic rotation must not be active during digitizing.
>
> The TNC also transfers the BLK FORM to the digitizing data file. For this purpose, the TNC enlarges the work-piece blank defined in the RANGE cycle by twice the value set in MP6310 (for measuring touch probes).

13.2 Programming Digitizing Cycles

▶ Press the TOUCH PROBE key.

▶ Select the desired digitizing cycle with the arrow keys.

▶ To confirm your selection, press the ENT key.

▶ Answer the dialog questions on the TNC screen: Enter the appropriate values with the keyboard and confirm each input with the ENT key. When all the required data have been entered, the TNC terminates the cycle definition automatically. For information on the individual input parameters refer to the cycle descriptions in this chapter.

Defining the digitizing range

The touch probe systems provide two cycles for defining the digitizing range. Cycle 5 RANGE defines a cuboid range within which the touch probe scans the contour. If you are using a measuring touch probe, you can also select a contour point table with Cycle 15 RANGE, in which the shape of the digitizing range is defined as a polygon.
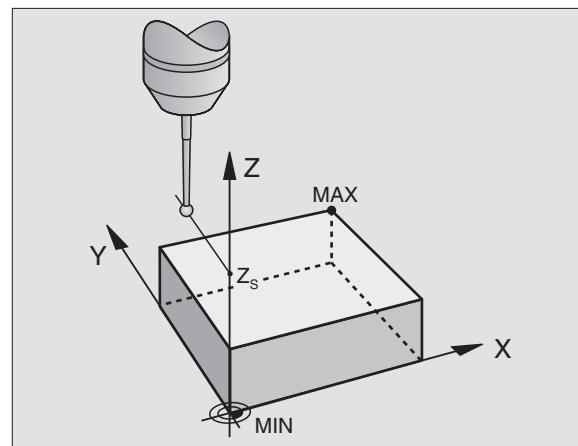
Defining a cuboid digitizing range
Similar to defining the workpiece blank, the digitizing range is programmed by entering the MIN and MAX point coordinates of the three main axes X, Y and Z (see figure at right).

▶ PGM NAME DIGITIZING DATA: Enter the name of the file in which the digitizing data is to be stored.

> In the screen menu for configuring the data interface, you must enter the complete directory path in which the TNC is to store the digitized data.

▶ TCH PROBE AXIS: Enter the touch probe axis.

▶ MIN. POINT RANGE: Lowest coordinates in the range to be digitized.

▶ MAX. POINT RANGE: Highest coordinates in the range to be digitized.

▶ CLEARANCE HEIGHT: Position in probe axis at which the stylus cannot collide with the model.

Resulting NC blocks

```
50  TCH  PROBE  5.0  RANGE
51  TCH  PROBE  5.1  PGM  NAME:  DATA
52  TCH  PROBE  5.2  Z  X+0  Y+0  Z+0
53  TCH  PROBE  5.3  X+10  Y+10  Z+20
54  TCH  PROBE  5.4  HEIGHT:  +  100
```

Defining the digitizing range in a specific shape (only available with measuring touch probes)
The digitizing range is defined by a contour point table generated in the POSITIONING WITH MDI mode of operation. You can either transfer the individual points by teach-in programming or move the touch probe around the model manually and have the TNC generate them automatically (see figure at right).

▶ PGM NAME DIGITIZING DATA: Enter the name of the file in which the digitizing data is to be stored.

☞ In the screen menu for configuring the data interface, you must enter the complete directory path in which the TNC is to store the digitized data.

▶ TCH PROBE AXIS: Enter the touch probe axis.

▶ PGM NAME RANGE DATA: Enter the name of the point table in which you have defined the range.

▶ MIN. POINT TCH PROBE AXIS: Lowest probe axis coordinate in the DIGITIZING range.

▶ MAX. POINT TCH PROBE AXIS: Highest probe axis coordinate in the DIGITIZING range.

▶ CLEARANCE HEIGHT: Position in probe axis at which the stylus cannot collide with the model.

Resulting NC blocks

```
50  TCH  PROBE  15.0  RANGE
51  TCH  PROBE  15.1  PGM  DIGIT.:  DATA
52  TCH  PROBE  15.2  Z  PGM  RANGE:  TAB1
53  TCH  PROBE  15.3  MIN:  +0  MAX:  +10  HIGHT:  +100
```
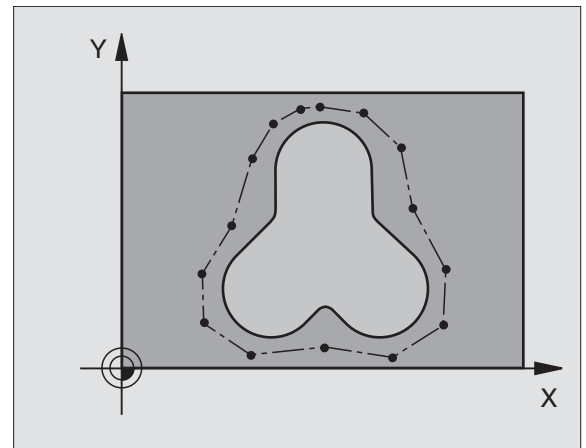
Point tables

When you are working with a measuring touch probe, you can generate point tables in the POSITIONING WITH MDI mode of operation. You can use these tables to define the digitizing range in a specific shape or to program a contour in a specific shape and then machine the contour with Cycle 30. This feature requires the "digitizing with measuring touch probe" software option from HEIDENHAIN.

You can transfer points in the following two ways:

- manually by TEACH-IN programming

- automatically by the TNC

☞ The TNC can store up to 893 points in a point table that is to be used for the digitizing range. You can have the TNC monitor the number of points stored in the point table by setting the TM:RANGE/CONTOUR DATA soft key to TM:RANGE.

The individual points are connected to form a progression of straight lines that define the digitizing range. The TNC automatically connects the last point in the table with the first point by a straight line.

To generate a point table

After inserting the measuring touch probe in the spindle and locking it mechanically, you can select a point table with the PNT soft key:

| PNT | Press the PNT soft key in the POSITIONING WITH MDI mode of operation. The TNC then displays soft-key rows with the following soft keys: |

| Function	Soft key
Transfer points manually	PROBE MAN
Transfer points automatically	PROBE AUTO
Select between digitizing range and contour	TM:RANGE CONTOUR DATA
Store/Do not store X coordinate	X OFF/ON
Store/Do not store Y coordinate	V OFF/ON
Store/Do not store Z coordinate	Z OFF/ON

▶ To select input of contour (TM:RANGE) or digitizing range (CON-TOUR DATA), set the TM:RANGE CONTOUR DATA soft key to the desired function.

If you want to transfer the points manually by teach-in programming, proceed as follows:

▶ To select manual transfer, press the PROBE MAN soft key. The TNC displays additional soft keys — see table at right.

▶ Select the feed rate at which the touch probe is to respond to a deflection by pressing the F soft key and entering the feed rate.

▶ Define whether or not you want the TNC to transfer the coordinates in specific axes by setting the X OFF/ON; Y OFF/ON and Z OFF/ON soft keys to the selected functions.

▶ To move the touch probe to the first point of the range to be digitized or to the first contour point, deflect the touch probe manually in the desired direction.

▶ Press the ACTUAL -POSITION-CAPTURE soft key. The TNC transfers the coordinates of the selected axes to the point table. The TNC uses the coordinates of the working plane only for defining the digitizing range.

▶ Move the touch probe to the next point and capture the actual position. Repeat this process until the whole range is defined.

If you want to have the TNC generate the points automatically, proceed as follows:

▶ To transfer points automatically, press the PROBE AUTO soft key. The TNC displays additional soft keys — see table at right.

▶ Select the feed rate at which the touch probe is to respond to a deflection by pressing the F soft key and entering the feed rate.

▶ Define the spacing between points at which the TNC transfers the points by pressing the "PROBE POINT INTERVAL" soft key and entering a value. After entry of the probe point interval, the TNC displays the START soft key.

▶ To move the touch probe to the first point of the range to be digitized or to the first contour point, deflect the touch probe manually in the desired direction.

▶ To start transfer, press the START soft key.

▶ Deflect the touch probe manually in the desired direction. The TNC transfers the coordinates in the programmed probe point interval.

▶ To terminate transfer, press the STOP soft key.

Function	Soft key
Feed rate at which the touch probe is to respond to a deflection	F
Store position in the point table with ACTUAL-POSITION-CAPTURE	

Function	Soft key
Feed rate at which the touch probe is to respond to a deflection	F
Define the probe point interval for automatic transfer	

13.3 Meander Digitizing

■ Triggering touch probe: Digitizing cycle 6 MEANDER

■ Measuring touch probe: Digitizing cycle 16 MEANDER

The MEANDER cycle scans and digitizes a 3-D surface in a back-and-forth series of parallel lines. This method is best suited for digitizing relatively flat surfaces. If you want to use the SUSA evaluation software to further process the data, you must use the MEANDER cycle.

Before starting the digitizing process, select an axis in the working plane. During digitizing, the touch probe moves in the positive direction of the selected axis, starting at the MIN point of the working plane. When it reaches the range limit, it moves by the line spacing in the positive direction of the other axis and then travels back along this line. At the other end of this line, the touch probe moves once again by the line spacing. This process is repeated until the entire range has been scanned.

Once the entire range has been scanned, the touch probe returns to the CLEARANCE HEIGHT.

When you are digitizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.

Starting position
■ MIN point coordinates from Cycle 5 RANGE or Cycle 15 RANGE in the working plane, probe axis coordinate = CLEARANCE HEIGHT.

■ Automatically move to the starting position:
First in the probe axis to CLEARANCE HEIGHT, then in the working plane.

Contour approach
The touch probe moves in the negative probe axis direction toward the model. When it makes contact, the TNC stores the position coordinates.

☞ Before the MEANDER cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters
The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(T)** apply to triggering touch probes:
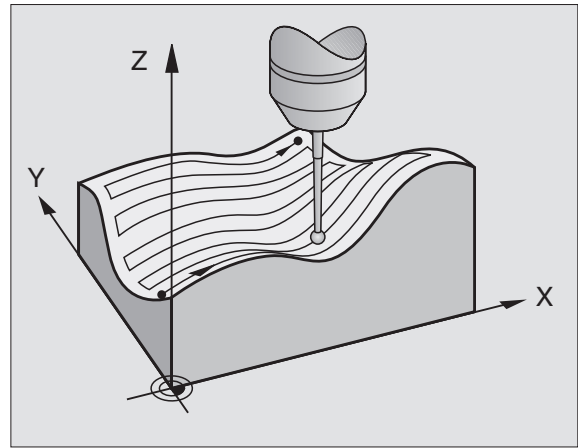
▶ LINE DIRECTION **(M, T)**: Coordinate axis in whose positive direction the touch probe moves from the first contour point.

▶ LIMIT IN NORMAL LINES DIRECTION **(T)**: Distance by which the touch probe lifts off the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the PROBE POINT INTERVAL. The smaller the ball-tip radius, the larger the LIMIT IN NORMAL LINES DIRECTION should be programmed.

▶ SCANNING ANGLE **(M)**: Traversing direction of the touch probe referenced to the LINE DIRECTION. Input range: –90° to +90°

▶ FEED RATE F **(M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ MIN. FEED RATE **(M)**: Probe feed rate for the first line. Input range: 1 to 3000 mm/min

▶ MIN. LINE SPACING **(M)**: If you enter an input value that is smaller than the programmed LINE SPACING, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing steep sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm

▶ LINE SPACING **(M, T)**: The offset by which the probe moves at the end of each line before scanning the next line. Input range: 0 to 20 mm

▶ MAX. PROBE POINT INTERVAL **(M, T)**: Maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 20 mm

▶ TOLERANCE **(M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This provides a high density of digitized positions for contours with sharply curved surfaces, while as few positions as possible are stored for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval. Input range: 0 to 0.9999 mm

▶ FEED RATE DECREASE AT EDGES **(M)**: Answer the dialog question with NO ENT. The TNC enters a value automatically.

☞ The FEED RATE DECREASE is only effective if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Resulting NC blocks for triggering touch probes

60	TCH PROBE 6.0 MEANDER
61	TCH PROBE 6.1 DIRECTN: X
62	TCH PROBE 6.2 TRAVEL: 0.5 L.SPAC: 0.2
	PP.INT: 0.5

Resulting NC blocks for measuring touch probes

60	TCH PROBE 16.0 MEANDER
61	TCH PROBE 16.1 DIRECTN: X
	ANGLE: +0
62	TCH PROBE 16.2 F1000 FMIN500
	MIN.L.SPAC: 0.2 L.SPAC: 0.5
	PP.INT: 0.5 TOL: 0.1 DIST: 2

13.4 Contour Line Digitizing

■ Triggering touch probe: Digitizing cycle 7 CONTOUR LINES

■ Measuring touch probe: Digitizing cycle 17 CONTOUR LINES

The CONTOUR LINES cycle scans a 3-D contour by circling around the model in a series of upwardly successive levels. This cycle is particularly useful for digitizing steep contours, such as the gating of injection moulding dies, or for digitizing a single contour line, such as the contour of a cam disk.

The touch probe starts scanning in the direction that you entered as STARTING PROBE AXIS AND DIRECTION at a constant height. When the touch probe has orbited the form and has returned to the first probe point, it then moves in the positive or negative direction of the probe axis by the value of the LINE SPACING. This process is then repeated for the rest of the range.

Once the entire range has been scanned, the touch probe returns to the CLEARANCE HEIGHT and the programmed starting point.

When you are digitizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.

Requirements for the scanning range
■ In the touch probe axis: The defined RANGE must be lower than the highest point of the 3-D model by at least the radius of the probe tip.

■ In the plane perpendicular to the touch probe: The defined RANGE must be larger than the 3-D model by at least the radius of the probe tip.

Starting position
■ Probe axis coordinate of the MIN point from Cycle 5 RANGE or Cycle 15 RANGE if the LINE SPACING is entered as a positive value.

■ Probe axis coordinate of the MAX point from Cycle 5 RANGE or Cycle 15 RANGE if the LINE SPACING is entered as a negative value.

■ Define the working plane coordinates in the CONTOUR LINES cycle.

■ Automatically approach the starting point:
First in the probe axis to CLEARANCE HEIGHT, then in the working plane.

Contour approach
The touch probe moves toward the surface in the programmed direction. When it makes contact, the TNC stores the position coordinates.
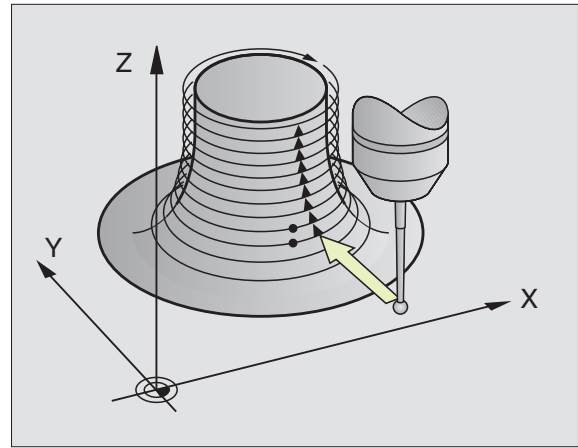
Before the CONTOUR LINES cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters

The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(T)** apply to triggering touch probes:

▶ TIME LIMIT **(M,T)**: Time within which the touch probe must orbit the model and reach the first touch point. You can define a maximum distance from the end point to the starting point with machine parameter MP 6390. If the entered time limit is exceeded, the TNC interrupts the digitizing process. Input range: 0 to 7200 seconds. The input value 0 means there is no time limit.

▶ STARTING POINT **(M,T)**: Coordinates of the starting position in the working plane.

▶ AXIS AND DIRECTION OF APPROACH **(M,T)**: Coordinate axis and direction in which the touch probe approaches the model.

▶ STARTING PROBE AXIS AND DIRECTION **(M,T)**: Coordinate axis and direction in which the probe begins scanning the model. Defining the direction at the same time determines the whether the subsequent machining operation is performed by up-cut or climb milling.

▶ FEED RATE F **(M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ MIN. FEED RATE **(M)**: Probe feed rate for the first contour line. Input range: 1 to 3000 mm/min

▶ MIN. LINE SPACING **(M)**: If you enter an input value that is smaller than the programmed LINE SPACING, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing flat sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm

▶ LINE SPACING AND DIRECTION **(M,T)**: The offset by which the touch probe moves to start a new contour line after completing the previous one. The algebraic sign determines the direction in which the probe is offset. Input range: –20 to +20 mm

If you only want to digitize a single contour line, enter zero for both the MIN. LINE SPACING and the LINE SPACING.

▶ MAX. PROBE POINT INTERVAL **(M,T)**: Maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners.
Input range: 0.02 to 20 mm

▶ TOLERANCE **(M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This ensures a high density of digitized positions for contours with sharply curved surfaces, while as few positions as possible are stored for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval.
Input range: 0 to 0.9999 mm

▶ FEED RATE REDUCTION AT EDGES **(M)**: Answer the dialog question with NO ENT. The TNC enters a value automatically.

The FEED RATE REDUCTION is effective only if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Resulting NC blocks for triggering touch probes

60	TCH PROBE 7.0 CONTOUR LINES
61	TCH PROBE 7.1 TIME: 0 X+0 Y+0
62	TCH PROBE 7.2 ORDER: Y– / X–
63	TCH PROBE 7.2 TRAVEL: 0.5 L.SPAC: +0.2
	PP.INT: 0.5

Resulting NC blocks for measuring touch probes

60	TCH PROBE 17.0 CONTOUR LINES
61	TCH PROBE 17.1 TIME: 0 X+0 Y+0
62	TCH PROBE 17.2 ORDER: Y– / X–
63	TCH PROBE 17.3 F1000 FMIN500
	MIN.L.SPAC: 0.2 L.SPAC: 0.5
	PP.INT: 0.5 TOL: 0.1 DIST: 2

13.5 Unidirectional Line Digitizing

- Triggering touch probe: Digitizing cycle 8 LINE

- Measuring touch probe: Digitizing cycle 18 LINE

The LINE cycle scans and digitizes a 3-D contour in a single direction in a series of parallel lines.

With measuring touch probes, this cycle is used mainly for digitizing with a rotary axis — see "Digitizing with a Rotary Axis."

With triggering touch probes, this cycle is best suited for digitizing relatively flat surfaces which are to be consistently up-cut or climb milled without prior evaluation of the digitized data.

In this cycle, the touch probe moves in the positive direction of the axis entered under DIRECTION. When the touch probe reaches the MAX coordinate of this axis, it returns to CLEARANCE HEIGHT. It then moves at rapid traverse back to the beginning of the next line and from there in the negative probe axis direction to the HEIGHT FOR FEED RATE DECREASE. From this point it continues traveling at the probing feed rate until it contacts the 3-D contour. This process is repeated until the entire range has been scanned. The paths of traverse in the LINE cycle are illustrated in the figure at lower right.

Once the entire range has been scanned, the touch probe returns to the CLEARANCE HEIGHT.

When you are digitizing with the measuring touch probe, the TNC stores positions at which sharp changes in direction have occurred. Up to 1000 positions per line can be stored. In the next line, the TNC automatically reduces the probe feed rate in the vicinity of such positions. This behavior improves the scanning results.

Starting position
- Positive or negative range limit of the programmed line direction (depending on the digitizing direction).

- MIN point coordinates from Cycle 5 RANGE or Cycle 15 RANGE in the working plane, probe axis coordinate = CLEARANCE HEIGHT.

- Automatically move to the starting position:
  First in the probe axis to CLEARANCE HEIGHT, then in the working plane.

Contour approach
The touch probe moves in the negative probe axis direction toward the model. When it makes contact, the TNC stores the position coordinates.
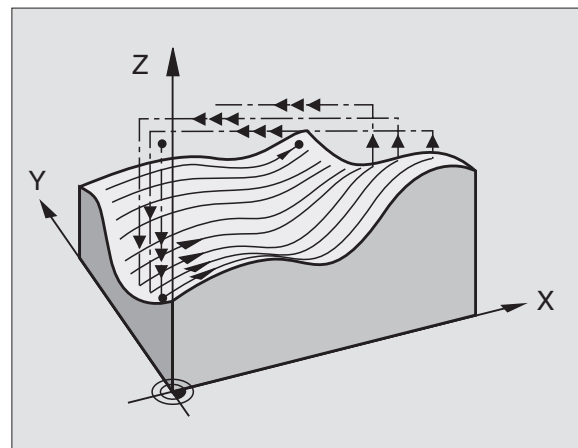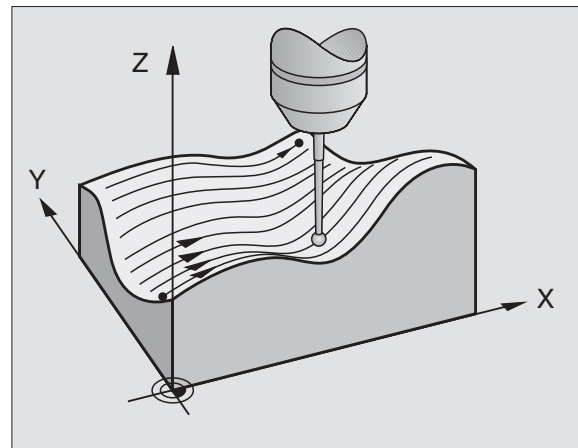
☞ Before the LINE cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters
The parameters marked with an **(M)** apply to measuring touch probes, those marked with an **(T)** apply to triggering touch probes:

▶ LINE DIRECTION **(M,T)**: Coordinate axis of the working plane parallel to which the touch probe scans the model. The digitizing direction at the same time defines whether the subsequent machining operation is performed by up-cut or climb milling.

▶ SCANNING ANGLE **(M)**: Traversing direction of the touch probe referenced to the LINE DIRECTION. By combining LINE DIRECTION and SCANNING ANGLE you can set the digitizing direction as desired. Input range: –90° to +90°

▶ HEIGHT FOR FEED RATE REDUCTION **(M,T)**: Probe axis coordinate at which the touch probe feed rate is reduced from rapid traverse to the probing feed rate at the beginning of each line. Input range: –99 999.9999 to +99 999.9999

▶ FEED RATE F **(M)**: Enter the digitizing speed. Input range: 1 to 3 000 mm/min. Note that the higher the digitizing speed, the less accurate the resulting data will be.

▶ MIN. FEED RATE **(M)**: Probe feed rate for the first line. Input range: 1 to 3 000 mm/min

▶ MIN. LINE SPACING **(M)**: If you enter an input value that is smaller than the programmed LINE SPACING, the TNC decreases the spacing between the scanned lines to the programmed minimum for digitizing steep sections of a contour. This ensures that the density of digitized positions remains constant even if complex surface structures are scanned. Input range: 0 to 20 mm

▶ LINE SPACING **(M,T)**: The offset by which the probe moves at the end of each line before scanning the next line. Input range: 0 to 20 mm

▶ MAX. PROBE POINT INTERVAL **(M,T)**: Maximum spacing between consecutive digitized positions. Input range: 0.02 to 20 mmm

▶ TOLERANCE **(M)**: The TNC stores only those digitized positions whose distance from a straight line defined by the last two probe points exceeds the programmed tolerance. This ensures a high density of digitized positions for contours with sharply curved surfaces, while as few positions as possible are stored for contours with flat surfaces. An input value of zero means that the digitized positions are output in the programmed probe point interval. Input range: 0 to 0.9999 mm

▶ FEED RATE DECREASE AT EDGES **(M)**: Distance in advance of steep edges, at which the TNC is to start reducing the digitizing feed rate.

☞ The FEED RATE DECREASE is only effective if the number of points at which the feed rate must be reduced does not exceed 1000 points in a digitizing line.

Resulting NC blocks for triggering touch probes

60	TCH PROBE 8.0 LINE
61	TCH PROBE 8.1 DIRECTN: X–
62	TCH PROBE 8.2 TRAVEL: 0.5 L.SPAC: 0.2
	PP.INT: 0.5

Resulting NC blocks for measuring touch probes

60	TCH PROBE 18.0 LINE
61	TCH PROBE 18.1 DIRECTN: X ANGLE: 0
	HEIGHT: 25
62	TCH PROBE 18.2 F1000 FMIN500
	MIN.L.SPAC: 0.2 L.SPAC: 0.5
	PP.INT: 0.5 TOL: 0.1 DIST: 2

13.6 Digitizing with a Rotary Axis

If you are using a triggering touch probe, the digitizing feature supports rotary axes with the cycles MEANDER (Cycle 6), LINE (Cycle 8) or CONTOUR LINE (Cycle 7). Regardless of the digitizing cycle used, the corresponding rotary axis must be entered in the RANGE cycle. The TNC interprets the rotary axis input as values in degrees.

If you are using a measuring touch probe, the digitizing feature supports rotary axes with cycle 18 LINE only. The rotary axis must be defined as a column axis.

Digitized data
The digitized data contains position information on the axes that are identified in the RANGE cycle.

The TNC does not generate a BLK FORM since rotary axes cannot be graphically simulated.

☞ The display mode of the rotary axis (i.e. whether the display reduces the values to below 360° or not) must be the same for digitizing as for milling.
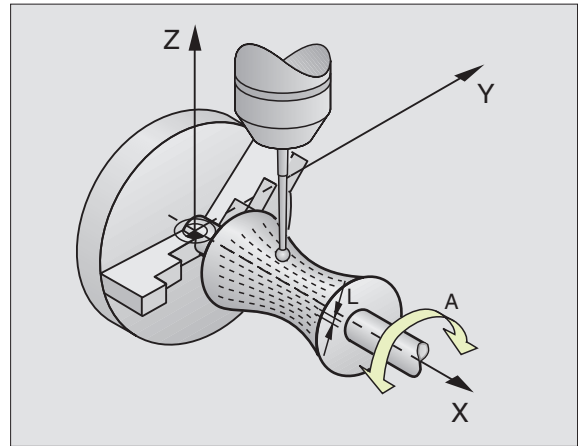
Measuring touch probe: LINE cycle with a rotary axis
If you enter a linear axis (e.g. X) in the input parameter LINE DIRECTION, the TNC moves the rotary axis (e.g. A) entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line (see the figures at right).

Resulting NC blocks

```
30   TCH PROBE 5.0 RANGE
31   TCH PROBE 5.1 PGMNAME: DATRND
32   TCH PROBE 5.2 Z X+0 A+0 Z+0
33   TCH PROBE 5.3 X+85 A+270 Z+25
34   TCH PROBE 5.4 HIGHT: 50
...
60   TCH PROBE 18.0 LINE
61   TCH PROBE 18.1 DIRECTN: X
     ANGLE: 0 HIGHT: 25
62   TCH PROBE 18.2 F1000
     MIN.L.SPAC: 0.2 L.SPAC: 0.5
     PP.INT: 0.5 TOL: 0.1 DIST: 2
```
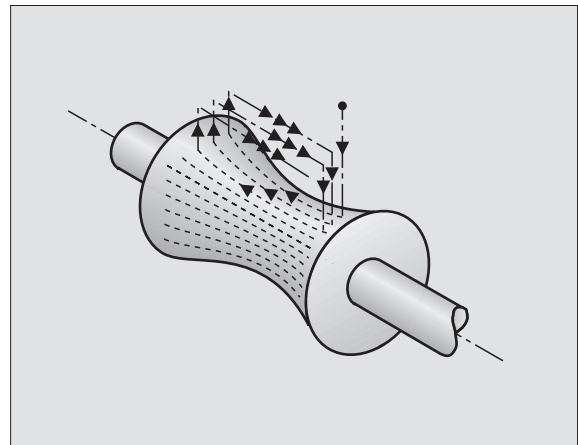
Triggering touch probe: MEANDER cycle with a rotary axis

If you enter a linear axis in the input parameter LINE DIRECTION, the TNC moves the rotary axis entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line. If the line direction is the linear axis X, and L.SPAC is in A, then the touch probe oscillates in the Z/X plane (see figure at upper right).

If you define the line direction as a rotary axis, the TNC moves the linear axis entered in the RANGE cycle by the line spacing L.SPAC as soon as it reaches the end of the line. If the line direction is the rotary axis A, and L.SPAC is in the linear axis X, then the touch probe oscillates in the Z/A plane (see figure at center right).

Resulting NC blocks

```
30   TCH PROBE 5.0 RANGE
31   TCH PROBE 5.1 PGMNAME: DATRND
32   TCH PROBE 5.2 Z X+0 A+0 Z+0
33   TCH PROBE 5.3 X+85 A+270 Z+65
34   TCH PROBE 5.4 HIGHT: 50
...
60   TCH PROBE 6.0 MEANDER
61   TCH PROBE 6.1 DIRECTNA
62   TCH PROBE 6.2 TRAVEL: 0.3 L.SPAC: 0.5 PP.INT: 0.5
```

CONTOUR LINES cycle with a rotary axis

Define the starting point in a linear axis and in a rotary axis. Define also the order and direction of axis approach. If the starting position lies in the X and C axes, then the touch probe oscillates in the X/C plane (see figure at lower right).

This method also works well for machines that have only two linear axes (e.g. Z/X) and one rotary axis (e.g. C).
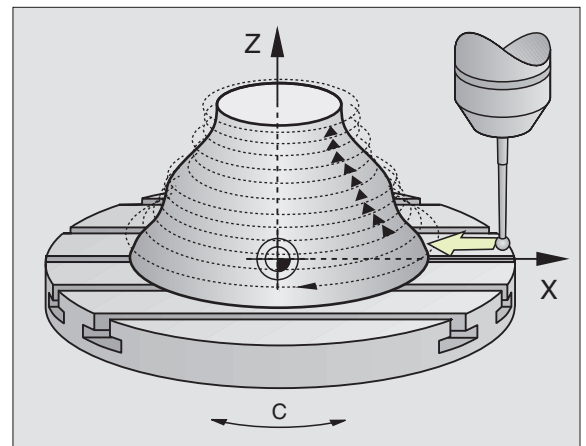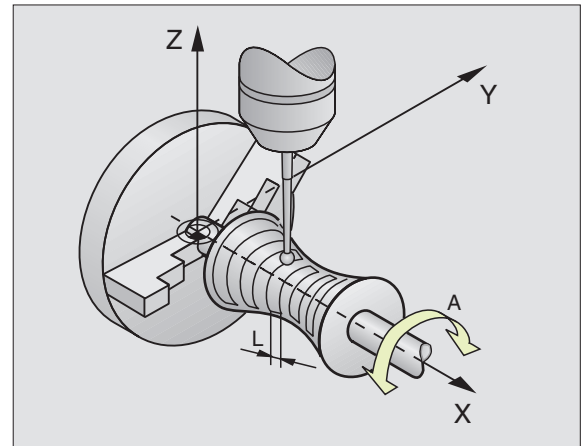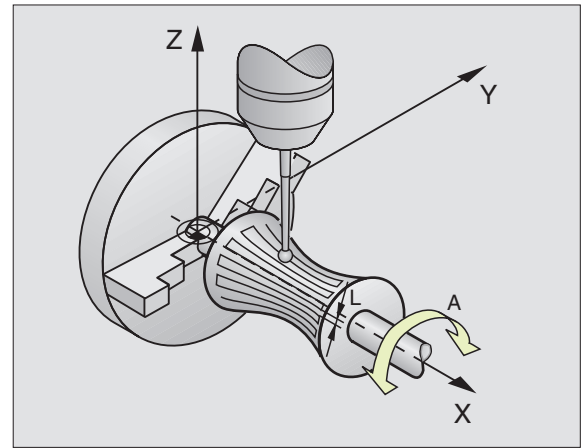
Resulting NC blocks

```
30   TCH PROBE 5.0 RANGE
31   TCH PROBE 5.1 PGMNAME: DATH
32   TCH PROBE 5.2 Z X–50 C+0 Z+0
33   TCH PROBE 5.3 X+50 C+360 Z+85
34   TCH PROBE 5.4 HIGHT: 50
...
60   TCH PROBE 7.0 CONTOUR LINES
61   TCH PROBE 7.1 TIME: 250 X+80 C+0
62   TCH PROBE 7.2 ORDER X–/C+
63   TCH PROBE 7.3 TRAVEL 0.3 L.SPAC: –0.5 PP.INT: 0.5
```

☞ The direction of rotation defined in ORDER is valid for all levels (lines). The direction at the same time defines whether the workpiece is then machined consistently in up-cut or in climb milling.

13.7 Using Digitized Data in a Part Program

Resulting NC blocks of a file containing data that were digitized with the CONTOUR LINES cycle.

`0    BEGIN PGM DATA MM`	Program name DATA is entered in the RANGE cycle
`1    BLK FORM 0.1 Z X-40 Y-20 Z+0`	Define workpiece blank: The size is determined by the TNC
`2    BLK FORM 0.2 X+40 Y+40 Z+25`	
`3    L Z+250 FMAX`	Clearance height in Z is entered in the RANGE cycle
`4    L X+0 Y-25 FMAX`	Starting point in X/Y is entered in the CONTOUR LINES cycle
`5    L Z+25`	Starting height in Z is entered in the CONTOUR LINES cycle and depends on the sign of the LINE SPACING
`6    L X+0.002 Y-12.358`	First digitized position
`7    L X+0.359 Y-12.021`	Second digitized position
`...`	
`253   L X+0.003 Y-12.390`	A contour line has been completed: probe has returned to the first digitized position
`254   L Z+24.5 X+0.017 Y-12.653`	
`...`	
`2597  L X+0.093 Y-16.390`	Last digitized position in the range
`2598  L X+0 Y-25 FMAX`	Back to the starting point in X/Y
`2599  L Z+250 FMAX`	Back to the clearance height in Z
`2600  END PGM DATA MM`	End of program

A part program can have up to 170 MB. This is the available memory capacity of the TNC's hard disk if no other programs are stored.

To execute a part program from digitized data, you can:

■ use fixed cycle 30, if several infeeds are required (only possible with data that were digitized in the MEANDER and LINE cycles, see section 8.7 "Cycles for Multipass Milling")

■ write an auxiliary program if you only need to finish-mill:

`0    BEGIN PGM MILLING MM`	
`1    TOOL DEF 1 L+0 R+4`	Tool definition: tool radius = stylus radius
`2    TOOL CALL 1 Z S4000`	Tool call
`3    L R0 F1500 M13`	Define feed rate for milling, spindle and coolant ON
`4    CALL PGM DATA`	Call digitized data
`5    END PGM MILLING MM`	

14

MOD Functions

14.1 Selecting, Changing and Exiting the MOD Functions

The MOD functions provide additional displays and input possibilities. The available MOD functions depend on the selected operating mode.

To select the MOD functions:
Call the mode of operation in which you wish to change the MOD function.

MOD ▶ To select the MOD functions, press the MOD key. The figures at right show typical screen menus in PROGRAMMING AND EDITING (figure at upper right), TEST RUN (figure at lower right) and in a machine operating mode (see figure on next page).

Depending on the selected mode of operation, you can make the following changes:

In the PROGRAMMING AND EDITING mode of operation:

■ Display NC software number

■ Display PLC software number

■ Enter code number

■ Set data interface

■ Machine-specific user parameters

■ HELP files (if provided)

In the TEST RUN mode of operation:

■ Display NC software number

■ Display PLC software number

■ Enter code number

■ Set data interface

■ Graphic display of the workpiece blank in the working area of the machine

■ Machine-specific user parameters

■ HELP files (if provided)

```
MANUAL       PROGRAMMING AND EDITING
OPERATION



CODE NUMBER              ████████

NC : SOFTWARE NUMBER     280462 01
PLC: SOFTWARE NUMBER
OPT:                     %00000011


 O⇁   RS 232   USER      HELP                      END
       RS 422  PARAMETER
       SETUP
```

```
MANUAL       TEST RUN
OPERATION



CODE NUMBER              ████████

NC : SOFTWARE NUMBER     280462 01
PLC: SOFTWARE NUMBER
OPT:                     %00000011


 O⇁   RS 232   DATUM   USER      HELP             END
       RS 422   SET    PARAMETER
       SETUP
```

In all other modes:

■ Display NC software number

■ Display PLC software number

■ Display code digits for installed options

■ Select position display

■ Unit of measurement (mm/inches)

■ Programming language for MDI

■ Select the axes for actual position capture

■ Axis traverse limits

■ Display datums

■ Display operating time

■ HELP files (if provided)

To change the MOD functions:

▶ Select the desired MOD function in the displayed menu with the arrow keys.

▶ Press the ENT key repeatedly until the desired function is highlighted or enter the appropriate numbers and confirm your entry with ENT.

To exit the MOD functions:

▶ Close the MOD functions with the END soft key or key.

```
MANUAL OPERATION                    PROGRAMMING
                                    AND EDITING


POSITION DISPLAY 1    ACTL.
POSITION DISPLAY 2    NOML.
CHANGE MM/INCH        MM
PROGRAM INPUT         HEIDENHAIN
AXIS SELECTION        %11111


NC : SOFTWARE NUMBER     280462 01
PLC: SOFTWARE NUMBER
OPT:                     %00000011



 POSITION/  AXIS           MACHINE
 INPUT PGM  LIMIT   HELP   TIME            END
```

14.2 Software Numbers and Option Numbers

The software numbers of the NC and PLC are displayed in the MOD function opening screen. Directly below them are the code numbers for the installed options (OPT:):

■ No option   OPT: 00000000

■ Digitizing option   OPT: 00000001

■ Digitizing and measuring touch probe options   OPT: 00000011

14.3 Code Number

A code number is required for access to the following function:

Function	Code number
To select user parameters	123

14.4 Setting the Data Interfaces

Press the soft key marked RS 232- / RS 422 - SETUP to call a menu for setting the data interfaces:

Setting the RS-232 interface

The mode of operation and baud rates for the RS-232 interface are entered in the upper left of the screen.

Setting the RS-422 interface

The mode of operation and baud rates for the RS-422 interface are entered in the upper right of the screen.

Setting the OPERATING MODE of the external device

☞ The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the operating modes FE2 and EXT.

Setting the BAUD RATE

You can set the BAUD-RATE (data transfer speed) from 110 to 115,200 baud.

```
MANUAL          PROGRAMMING AND EDITING
OPERATION

RS232 INTERFACE      RS422 INTERFACE

MODE OF OP.: LSV-2   MODE OF OP.: LSV-2
BAUD RATE           BAUD RATE
FE   :   9600       FE   :   9600
EXT1 :   9600       EXT1 :   9600
EXT2 :   9600       EXT2 :   9600
LSV-2:   9600       LSV-2:   9600

ASSIGN:

PRINT     :   TNC:\SCREENS\NEUEBA
PRINT-TEST :

  ⊶   | RS 232 | USER    | HELP |  |  | END
      | RS 422 | PARAMETER|
      | SETUP  |
```

External device	Mode	Symbol
HEIDENHAIN floppy disk units FE 401 B FE 401 with program no. 230 626 03 or higher	FE1 FE1	🖫
HEIDENHAIN FE 401 floppy disk unit with program no. below 230 626 02	FE2	🖫
PC with HEIDENHAIN data transfer software TNC. EXE	FE1	🖫
Non-HEIDENHAIN devices such as printers, readers, tape punchers, PC without TNC. EXE	EXT1, EXT2	⌦
PC with HEIDENHAIN software TNC REMOTE for remote operation	LSV2	🖥

ASSIGN

This function sets the destination for the transferred data.

Applications:

- Transferring values with Q parameter function FN15

- Transferring values with Q parameter function FN16

- Path on the TNC's hard disk in which the digitized data are stored

The TNC mode of operation determines whether the PRINT or PRINT TEST function is used:

TNC mode of operation	Transfer function
PROGRAM RUN, SINGLE BLOCK	PRINT
PROGRAM RUN, FULL SEQUENCE	PRINT
TEST RUN	PRINT TEST

You can set PRINT and PRINT TEST as follows:

Function	Path
Transfer data with FN15/FN16 via RS-232	RS232:\....
Transfer data with FN15/FN16 via RS-422	RS422:\....
Save data to the TNC's hard disk	TNC:\....
Save data in directory in which the program with FN15/FN16 or the program with the digitizing cycles is located	- vacant -

File names:

Data	Mode of operation	File name
Digitizing data	PROGRAM RUN	Defined in the RANGE cycle
Values with FN15	PROGRAM RUN	%FN15RUN.A
Values with FN15	TEST RUN	%FN15SIM.A
Values with FN16	PROGRAM RUN	%FN16RUN.A
Values with FN16	TEST RUN	%FN16SIM.A

14.5 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 user parameters. For more detailed information on user parameters, refer to your machine operating manual.

14.6 Showing the Workpiece in the Working Space

This MOD function enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the TEST RUN mode of operation. This function is activated with the DATUM SET soft key.

The TNC displays the working space, various windows containing coordinate information, and soft keys with which you can change the display.

Available traversing range/datums, referenced to the displayed workpiece blank:

1 Working space

2 Size of the blank

3 Coordinate system

4 Workpiece blank with orthogonal projections, working space

To show the position of the workpiece blank referenced to the datum, press the soft key marked with the machine symbol.

If the workpiece blank is located outside the working space 4, you can use the datum soft keys to move it within the graphic in such a way that the whole blank is located within the working space. You must subsequently move the datum in the MANUAL OPERATION mode by the same distance.
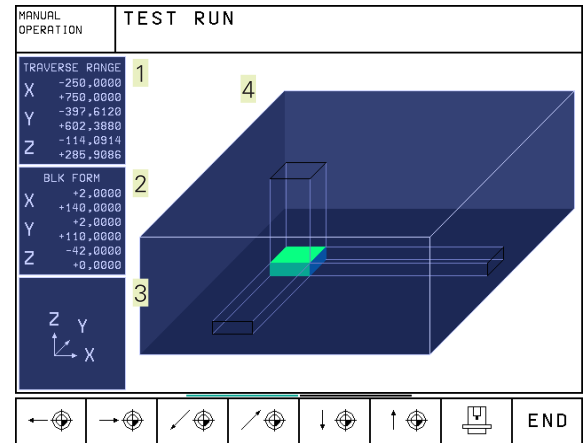
Overview of functions

Function	Soft key
Move workpiece blank to the left (graphically)	← ⊕
Move workpiece blank to the right (graphically)	→ ⊕
Move workpiece blank forward (graphically)	⟋ ⊕
Move workpiece blank backward (graphically)	⟋ ⊕
Move workpiece blank upward (graphically)	↑ ⊕
Move workpiece blank downward (graphically)	↓ ⊕
Show workpiece blank referenced to the set datum	⊡
Show the entire traversing range referenced to the displayed workpiece blank	⊢——⊣
Show the machine datum in the working space	M91 ⊕
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	M92 ⊕
Show the workpiece datum in the working space	⊕
Enable (ON) or disable (OFF) work space monitoring during test run	⊢——⊣ OFF/ON

14.7 Position Display Types

In the MANUAL OPERATION mode and in the program run modes of operation, you can select the type of coordinates to be displayed.

The figure at right shows the different tool positions:

1 Starting position

2 Target position of the tool

3 Workpiece datum

4 Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position; the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; the actual position as referenced to the machine datum	R E F
Distance remaining to the programmed position; difference between actual and target positions	DIST.
Servo lag; difference between nominal and actual positions	LAG
Deflection of the measuring touch probe	DEFL.

With the MOD function POSITION DISPLAY 1 you can select the position display in the status display. With POSITION DISPLAY 2 you can select the position display in the additional status display.

14.8 Unit of Measurement

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

■ To select the metric system (e.g. X = 15.789 mm) set the CHANGE MM/INCHES function to MM. The value is displayed with 3 digits after the decimal point.

■ To select the inch system (e.g. X = 0.6216 inch) set the CHANGE MM/INCHES function to INCHES. The value is displayed to 4 decimal places.
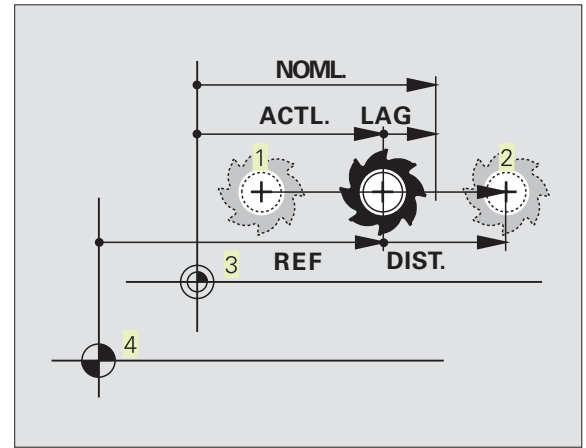
14.9 Programming Language for $MDI

The PROGRAM INPUT mod function lets you decide whether to program the $MDI file in HEIDENHAIN conversational dialog or in ISO format:

■ To program the $MDI.H file in conversational dialog,
  set the PROGRAM INPUT function to HEIDENHAIN.

■ To program the $MDI.I file according to ISO,
  set the PROGRAM INPUT function to ISO.

14.10 Selecting the Axes for Generating L Blocks

The AXIS SELECTION input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the ACTUAL-POSITION-CAPTURE soft key. The axes are selected by bit-oriented definition similar to programming the machine parameters:

AXIS SELECTION   %11111   Transfer the X, Y, Z, IV, and V axes

AXIS SELECTION   %01111   Transfer the X, Y, Z, and IV axes

AXIS SELECTION   %00111   Transfer the X, Y, and Z axes

AXIS SELECTION   %00011   Transfer the X and Y axes

AXIS SELECTION   %00001   Transfer the X axis

14.11 Axis Traverse Limits, Datum Display

The AXIS LIMIT mod function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application:
to protect an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the AXIS LIMIT mod function. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum.

Working without additional traverse limits
To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99 999 mm) as the AXIS LIMIT.
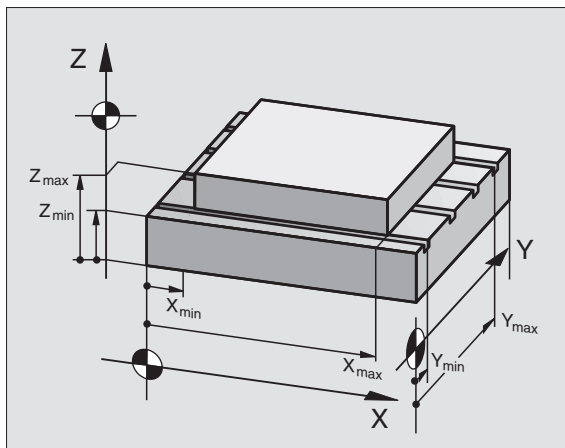
To find and enter the maximum traverse:

▶ Set the POSITION DISPLAY mod function to REF.

▶ Move the spindle to the positive and negative end positions of the X, Y and Z axes.

▶ Write down the values, including the algebraic sign.

▶ To select the MOD functions, press the MOD key.

| AXIS LIMIT | ▶ To enter the traverse limits, press the AXIS LIMIT soft key and enter the values that you wrote down as LIMITS in the corresponding axes. |

      ▶ To exit the MOD functions, press the END soft key.

☞ The tool radius is not automatically compensated in the axis traverse limit value.

The traverse range limits and software limit switches become active as soon as the reference points are traversed.

```
MANUAL OPERATION                    PROGRAMMING
                                    AND EDITING

LIMITS:
    X-  -500            X+  +500
    Y-  -500            Y+  +500
    Z-  +0              Z+  +400
    C-  +0              C+  +360
    B-  -90             B+  +90

DATUM POINTS:
    X   +250
    Y   +102,388
    Z   -114,0914
    C   +30
    B   +90

POSITION/  AXIS          MACHINE
INPUT PGM  LIMIT   HELP  TIME            END
```

Datum display

The values shown at the lower left of the screen are the manually set datums referenced to the machine datum. They cannot be changed in the menu.

14.12 HELP Files

Help files can aid you in situations in which you need clear instructions before you can continue (for example, to retract the tool after an interruption of power). The miscellaneous functions may also be explained in a help file. The figure at right shows the screen display of a help file.

✎ HELP files are not provided on every machine. Your machine tool builder can provide you with further information on this feature.

To call HELP files:

▶ Select the MOD functions with the MOD key.

| HELP | ▶ To select the last active HELP file, press the HELP soft key. |

      ▶ Call the file manager and select a different help file, if necessary.

```
PROGRAMMING AND EDITING             PROGRAMMING
                                    AND EDITING

FILE: TOOLCHANGER       LINE: 1   COLUMN: 1   INSERT
COMMANDS FOR THE TOOL CHANGER !!!


#0001 CHAIN FORWARD
#0002 CHAIN BACKWARD
[END]




ACTL.     X    +25,3684    Y   -250,3600
          Z    -25,0000    B   +331,0000
          C    +12,5000
T                          F  0        M 5/9

INSERT    MOVE   MOVE   PAGE   PAGE   BEGIN  END    FIND
OVERWRITE WORD   WORD                 TEXT   TEXT
          >>     <<
```

14.13 Operating Time

⏚ The machine tool builder can provide further operating
time displays. Refer to your machine tool manual for
more information on operating times.

The MACHINE TIME soft key enables you to show different operat-
ing time displays:

Operating time	Meaning
CONTROL ON	Operating time of the control since commissioning
MACHINE ON	Operating time of the machine tool since commissioning
PROGRAM RUN	Operating time for program-controlled operation since commissioning

```
MANUAL OPERATION                    PROGRAMMING
                                    AND EDITING


CONTROL ON        =    797:50:39
MACHINE ON        =    109:51:38
PROGRAM RUN       =     55:33:20










                                            END
```

15

Tables and Overviews

15.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements.

Some examples of user parameters are:

■ Dialog language

■ Interface behavior

■ Traversing speeds

■ Sequence of machining

■ Effect of overrides

Input possibilities for machine parameters

Machine parameters can be programmed as

■ **Decimal numbers**
  Enter only the number

■ **Pure binary numbers**
  Enter a percent sign (%) before the number

■ **Hexadecimal numbers**
  Enter a dollar sign ($) before the number

Example:
Instead of the decimal number 27 you can also enter the binary number %11011 or the hexadecimal number $1B.

The individual machine parameters can be entered in the different number systems.

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.

☞ The MOD functions also include machine-specific user parameters (USER PARAMETERS).

External data transfer

Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device	**MP5020.x**
	7 data bits (ASCII code, 8th bit = parity): **+0**
	8 data bits (ASCII code, 9th bit = parity): **+1**
	Block Check Character (BCC) any: **+0**
	Block Check Character (BCC) control character not permitted: **+2**
	Transmission stop through RTS active: **+4**
	Transmission stop through RTS inactive: **+0**
	Transmission stop through DC3 active: **+8**
	Transmission stop through DC3 inactive: **+0**
	Character parity even: **+0**
	Character parity odd: **+16**
	Character parity not desired: **+0**
	Character parity desired: **+32**
	$1\frac{1}{2}$ stop bits: **+0**
	2 stop bits: **+64**
	1 stop bit: **+128**
	1 stop bit: **+192**

Example:

Use the following setting to adjust the TNC interface EXT2 (MP 5020.1) to an external non-HEIDENHAIN device:

8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits

Input for **MP 5020.1**: 1+0+8+0+32+64 = **105**

Interface type for EXT1 (5030.0) and EXT2 (5030.1)	**MP5030.x**
	Standard transmission: **0**
	Interface for blockwise transfer: **1**

3-D touch probes and digitizing

Select touch probe	**MP6200** Triggering touch probe: **0** Measuring touch probe: **1**
Select signal transmission	**MP6010** Touch probe with cable transmission: **0** Touch probe with infrared transmission: **1**
Probing feed rate for triggering touch probes	**MP6120** **80** to **3000** [mm/min]
Maximum traverse to first probe point	**MP6130** **0.001** to **99 999.9999** [mm]
Safety clearance to probing point during automatic measurement	**MP6140** **0.001** to **99 999.9999** [mm]
Rapid traverse for triggering touch probes	**MP6150** **1** to **300 000** [mm/min]
Measure center misalignment of the stylus when calibrating a triggering touch probe	**MP6160** No 180° rotation of the 3-D touch probe during calibration: **0** M function for 180° rotation of the 3-D touch probe during calibration: **1** to **88**
Reserved	**MP6300**
Infeed of the stylus when digitizing with a measuring touch probe	**MP6310** **0.1** to **2.0000** [mm] (recommended input value: 1 mm)
Measure center misalignment of the stylus when calibrating a measuring touch probe	**MP6321** Measure center misalignment: **0** Do not measure center misalignment: **1**

Assign touch probe axis to machine axis for a measuring touch probe

☞ Ensure that the touch probe axes are correctly assigned to the machine axes. Wrong assignment could lead to a stylus break.

	MP6322.0 Machine **X** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**
	MP6322.1 Machine **Y** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**
	MP6322.2 Machine **Z** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**

Maximum stylus deflection on measuring touch probes	**MP6330** **0.1** to **4.0000** [mm]
Feed rate for positioning measuring touch probes at MIN point and approaching contour	**MP6350** **10** to **3 000** [mm/min]
Probe feed rate for measuring touch probes	**MP6360** **10** to **3 000** [mm/min]
Rapid traverse for measuring touch probes in the probe cycle	**MP6361** **10** to **3 000** [mm/min]

Feed rate reduction when the stylus of a measuring touch probe is deflected to the side

The TNC decreases the feed rate according to a preset characteristic curve. The minimum input value is 10% of the programmed digitizing feed rate.

MP6362
Feed rate reduction not active: **0**
Feed rate reduction active: **1**

Radial acceleration during digitizing with measuring touch probes

MP6370 enables you to limit the feed rate of the TNC for circular movements during digitizing. Circular movements are caused, for example, by sharp changes of direction.

As long as the programmed digitizing feed rate is less than the feed rate calculated with MP6370, the TNC will move at the programmed feed rate. Determine the appropriate value for your requirements by trial and error.

MP6370
0.001 to **5.000** [m/s^2] (recommended input value: 0.1)

Target window for digitizing contour lines with measuring touch probes

When you are digitizing contour lines the individual contour lines do not end exactly in their starting points.

With machine parameter MP6390 you can define a square target window within which the end point must lie after the touch probe has orbited the model. Enter half the side length of the target window for the side length.

MP6390
0.1 to **4.0000** [mm]

Radius measurement with the TT 120 touch probe: Probing direction

MP6505

Positive probing direction in the angle reference axis (0° axis): **0**

Positive probing direction in the +90° axis: **1**

Negative probing direction in the angle reference axis (0° axis): **2**

Negative probing direction in the +90° axis: **3**

Probing feed rate for second measurement with the TT 120, stylus shape, corrections in TOOL.T

MP6507

Calculate feed rate for second measurement with TT 120, with constant tolerance: **+0**

Calculate feed rate for second measurement with TT 120, with variable tolerance: **+1**

Constant feed rate for second measurement with TT 120: **+2**

Maximum permissible error or measurement for measuring rotating tools with the TT 120

Required for calculating the probing feed rate in connection with MP6570

MP6510

0.001 to **0.999** [mm] (recommended input value: 0.005 mm)

Feed rate for probing a tool with the TT 120 during standstill of the tool

MP6520

10 to **3 000** [mm/min]

Radius measurement with the TT 120: Distance from lower edge of tool to upper edge of stylus

MP6530

0.0001 to **9 999.9999** [mm]

Clearance zone around the stylus of the TT 120 for pre-positioning

MP6540

0.001 to **99 999.999** [mm]

Rapid traverse for TT 120 in the probe cycle

MP6550

10 to **10 000** [mm/min]

M function for spindle orientation when measuring individual teeth

MP6560

0 to **88**

Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool

Required for calculating rpm and probe feed rate

MP6570

1.000 to **120.000** [m/min]

TNC displays, TNC editor

Programming station	**MP7210**
	TNC with machine: **0**
	TNC as programming station with active PLC: **1**
	TNC as programming station with inactive PLC: **2**
Acknowledgment of POWER INTERRUPTED after switch-on	**MP7212**
	Acknowledge with key: **0**
	Acknowledge automatically: **1**
ISO programming: Block number increment	**MP7220**
	0 to **150**

Inhibit particular file type

☞ If a particular file type is inhibited, the TNC will erase all files of this type.

	MP7224.0
	Do not inhibit file types: **+0**
	Inhibit HEIDENHAIN programs: **+1**
	Inhibit ISO programs: **+2**
	Inhibit tool tables: **+4**
	Inhibit datum tables: **+8**
	Inhibit pallet tables: **+16**
	Inhibit text files: **+32**
Inhibit the editor for certain file types	**MP7224.1**
	Do not inhibit editor: **+0**
	Inhibit editor for
	■ HEIDENHAIN programs: **+1**
	■ ISO programs: **+2**
	■ Tool tables: **+4**
	■ Datum tables: **+8**
	■ Pallet tables: **+16**
	■ Text files: **+32**
Configure pallet files	**MP7226.0**
	Pallet table inactive: **0**
	Number of pallets per pallet table: **1** to **255**
Configure datum files	**MP7226.1**
	Datum table inactive: **0**
	Number of datums per datum table: **1** to **255**
Program length for program check	**MP7229.0**
	Blocks **100** to **9 999**
Program length up to which FK blocks are permitted	**MP7229.1**
	Blocks **100** to **9 999**

Dialog language	**MP7230**
	English: **0**
	German: **1** Portuguese: **6**
	Czech: **2** Swedish: **7**
	French: **3** Danish: **8**
	Italian: **4** Finnish: **9**
	Spanish: **5** Dutch: **10**
Internal clock of the TNC	**MP7235**
	Universal time (Greenwich Mean Time): **0**
	Central European Time (CET): **1**
	Central European Summertime: **2**
	Time difference to universal time: **-23** to **+23** [hours]
Configure tool tables	**MP7260**
	Inactive: **0**
	Number of tools per tool table: **1** to **254**
Configure pocket tables	**MP7261**
	Inactive: **0**
	Number of pockets per pocket table: **1** to **254**

Configure tool table (To omit from the table: enter 0);
Column number in the tool table for

MP7266.0	Tool name — NAME: **0** to **24**
MP7266.1	Tool length — L: **0** to **24**
MP7266.2	Tool radius — R: **0** to **24**
MP7266.3	Tool radius 2 — R2: **0** to **24**
MP7266.4	Oversize length — DL: **0** to **24**
MP7266.5	Oversize radius — DR: **0** to **24**
MP7266.6	Oversize radius 2 — DR2: **0** to **24**
MP7266.7	Tool locked — TL: **0** to **24**
MP7266.8	Replacement tool — RT: **0** to **24**
MP7266.9	Maximum tool life — TIME1: **0** to **24**
MP7266.10	Maximum tool life for TOOL CALL — TIME2: **0** to **24**
MP7266.11	Current tool life — CUR. TIME: **0** to **24**
MP7266.12	Tool comment — DOC: **0** to **24**
MP7266.13	Number of teeth — CUT.: **0** to **24**
MP7266.14	Tolerance for wear detection in tool length — LTOL: **0** to **24**
MP7266.15	Tolerance for wear detection in tool radius — RTOL: **0** to **24**
MP7266.16	Cutting direction — DIRECT.: **0** to **24**
MP7266.17	PLC status — PLC: **0** to **24**
MP7266.18	Offset of the tool in the tool axis in addition to MP6530 — TT:L-OFFS: **0** to **24**
MP7266.19	Offset of the tool between stylus center and tool center — TT:R-OFFS: **0** to **24**
MP7266.20	Tolerance for break detection in tool length — LBREAK.: **0** to **24**
MP7266.21	Tolerance for break detection in tool radius — RBREAK: **0** to **24**
MP7266.22	Tooth length (Cycle 22) — LCUTS: **0** to **24**
MP7266.23	Maximum plunge angle (Cycle 22) — ANGLE.: **0** to **24**

Configure pocket table; column number of the data in the tool table for (To omit from the table: enter 0)	**MP7267.0** Tool number — T: **0** to **5**
	MP7267.1 Special tool — ST: **0** to **5**
	MP7267.2 Fixed pocket — F: **0** to **5**
	MP7267.3 Pocket locked — L: **0** to **5**
	MP7267.4 PLC status — PLC: **0** to **5**
MANUAL OPERATION mode: Display feed rate	**MP7270** Display feed rate F only if an axis direction button is pressed: **0** Display feed rate F even if no axis direction button is pressed (feed rate of the "slowest" axis): **1**
Decimal character	**MP7280** The decimal character is a comma: **0** The decimal character is a point: **1**
Position display in the tool axis	**MP7285** Display is referenced to the tool datum: **0** Display in the tool axis is referenced to the tool face: **1**
Display step for the X axis	**MP7290.0** 0.1 mm: **0** 0.05 mm: **1** 0.001 mm: **4** 0.01 mm: **2** 0.0005 mm: **5** 0.005 mm: **3** 0.0001 mm: **6**
Display step for the Y axis	**MP7290.1** 0.1 mm: **0** 0.05 mm: **1** 0.001 mm: **4** 0.01 mm: **2** 0.0005 mm: **5** 0.005 mm: **3** 0.0001 mm: **6**
Display step for the Z axis	**MP7290.2** 0.1 mm: **0** 0.05 mm: **1** 0.005 mm: **3** 0.01 mm: **2** 0.001 mm: **4**

Display step for the IV axis	**MP7290.3** 0.1 mm: **0** 0.05 mm: **1** 0.001 mm: **4** 0.01 mm: **2** 0.0005 mm: **5** 0.005 mm: **3** 0.0001 mm: **6**
Display step for the V axis	**MP7290.4** 0.1 mm: **0** 0.05 mm: **1** 0.001 mm: **4** 0.01 mm: **2** 0.0005 mm: **5** 0.005 mm: **3** 0.0001 mm: **6**
Disable datum setting	**MP7295** Do not disable datum setting: **+0** Disable datum setting in the X axis: **+1** Disable datum setting in the Y axis: **+2** Disable datum setting in the Z axis: **+4** Disable datum setting in the IV axis: **+8** Disable datum setting in the V axis: **+16**
Disable datum setting with the orange axis keys	**MP7296** Do not inhibit datum setting: **0** Disable datum setting with the orange axis keys: **1**
Reset status display, Q parameters and tool data	**MP7300** Reset them all when a program is selected: **0** Reset them all when a program is selected and with M02, M30, END PGM: **1** Reset only status display and tool data when a program is selected: **2** Reset only status display and tool data when a program is selected and with M02, M30, END PGM: **3** Reset status display and Q parameters when a program is selected: **4** Reset status display and Q parameters when a program is selected and with M02, M30, END PGM: **5** Reset status display when a program is selected: **6** Reset status display when a program is selected and with M02, M30, END PGM: **7**
Graphic display mode	**MP7310** Projection in three planes according to ISO 6433, projection method 1: **+0** Projection in three planes according to ISO 6433, projection method 2: **+1** Do not rotate coordinate system for graphic display: **+0** Rotate coordinate system for graphic display by 90°: **+2** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the previous datum: **+0** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the new datum: **+4** Do not show cursor position during projection in three planes: **+0** Show cursor position during projection in three planes: **+8**

Graphic simulation without programmed tool axis: Tool radius	**MP7315**	
	0 to **99 999.9999** [mm]	
Graphic simulation without programmed tool axis: Penetration depth	**MP7316**	
	0 to **99 999.9999** [mm]	
Graphic simulation without programmed tool axis: M function for start	**MP7317.0**	
	0 to **88** (0: Function inactive)	
Graphic simulation without programmed tool axis: M function for end	**MP7317.1**	
	0 to **88** (0: Function inactive)	

Screen saver

Enter the time after which the TNC should start the screen saver

MP7392
0 to **99** [min] (0: Function inactive)

Machining and program run

Cycle 17: Oriented spindle stop at beginning of cycle

MP7160
Spindle orientation: **0**
No spindle orientation: **1**

Effect of Cycle 11 SCALING FACTOR

MP7410
SCALING FACTOR effective in 3 axes: **0**
SCALING FACTOR effective in the working plane only: **1**

Tool data in programmable probe cycle TOUCH PROBE 0

MP7411
Overwrite current tool data by the calibrated data from the 3-D touch probe system: **0**
Current tool data are retained: **1**

SL cycles	**MP7420**
	Mill channel around the contour — clockwise for islands and counterclockwise for pockets: **+0**
	Mill channel around the contour — clockwise for pockets and counterclockwise for islands: **+1**
	First mill the channel, then rough out the contour: **+0**
	First rough out the contour, then mill the channel: **+2**
	Combine compensated contours: **+0**
	Combine uncompensated contours: **+4**
	Complete one process for all infeeds before switching to the other process: **+0**
	Mill channel and rough-out for each infeed depth before continuing to the next depth: **+8**
	The following note applies to the Cycles 6, 15, 16, 21, 22, 23, and 24:
	At the end of the cycle, move the tool to the position that was last programmed before the cycle call: **+0**
	At the end of the cycle, retract the tool in the tool axis only: **+16**
Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor	**MP7430** **0.1** to **1.414**
Permissible distance by which an end point can be off the path of a perfect circle	**MP7431** **0.0001** to **0.016** [mm]
Behavior of M functions	**MP7440**
	Program stop with M06: **+0**
	No program stop with M06: **+1**
	No cycle call with M89: **+0**
	Cycle call with M89: **+2**
	Program stop with M functions: **+0**
	No program stop with M functions: **+4**
	k_v factors cannot be switched through M105 and M106: **+0**
	k_v factors can be switched through M105 and M106: **+8**
	Reduce the feed rate in the tool axis with M103 F.. , function inactive: **+0**
	Reduce the feed rate in the tool axis with M103 F.. , function active: **+16**

> The k_v factors for position loop gain are set by the machine tool builder. Your machine manual provides more detailed information on this subject.

Corners whose angles are less than the entered value will be machined at a decelerated feed rate (corner with R0, inside corner also radius-compensated)

This feature works both during operation with servo lag as well as with velocity feedforward.

	MP7460 **0.0000** to **179.9999** [°]

**Maximum contouring speed at
a feed rate override setting of 100%
in the program run modes**

MP7470
0 to **99 999** [mm/min]

**Datums from a datum table
are referenced to the**

MP7475
Workpiece datum: **0**
Machine datum: **1**

Electronic handwheels

Handwheel type

MP7640
Machine without handwheel: **0**
HR 330 with additional keys — the handwheel keys for traverse direction and rapid traverse are evaluated by the NC: **1**
HR 130 without additional keys: **2**
HR 330 with additional keys — the handwheel keys for traverse direction and rapid traverse are evaluated by the PLC: **3**
HR 332 with twelve additional keys: **4**
Multi-axis handwheel with additional keys: **5**
HR 410 with auxiliary functions: **6**

Interpolation factor

MP7641
Interpolation factor is entered on the keyboard: **0**
Interpolation factor is set by the PLC: **1**

**Machine parameters that can be set for
the handwheel by the machine tool builder**

MP 7645.0	**0** to **255**
MP 7645.1	**0** to **255**
MP 7645.2	**0** to **255**
MP 7645.3	**0** to **255**
MP 7645.4	**0** to **255**
MP 7645.5	**0** to **255**
MP 7645.6	**0** to **255**
MP 7645.7	**0** to **255**

15.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface

HEIDENHAIN devices
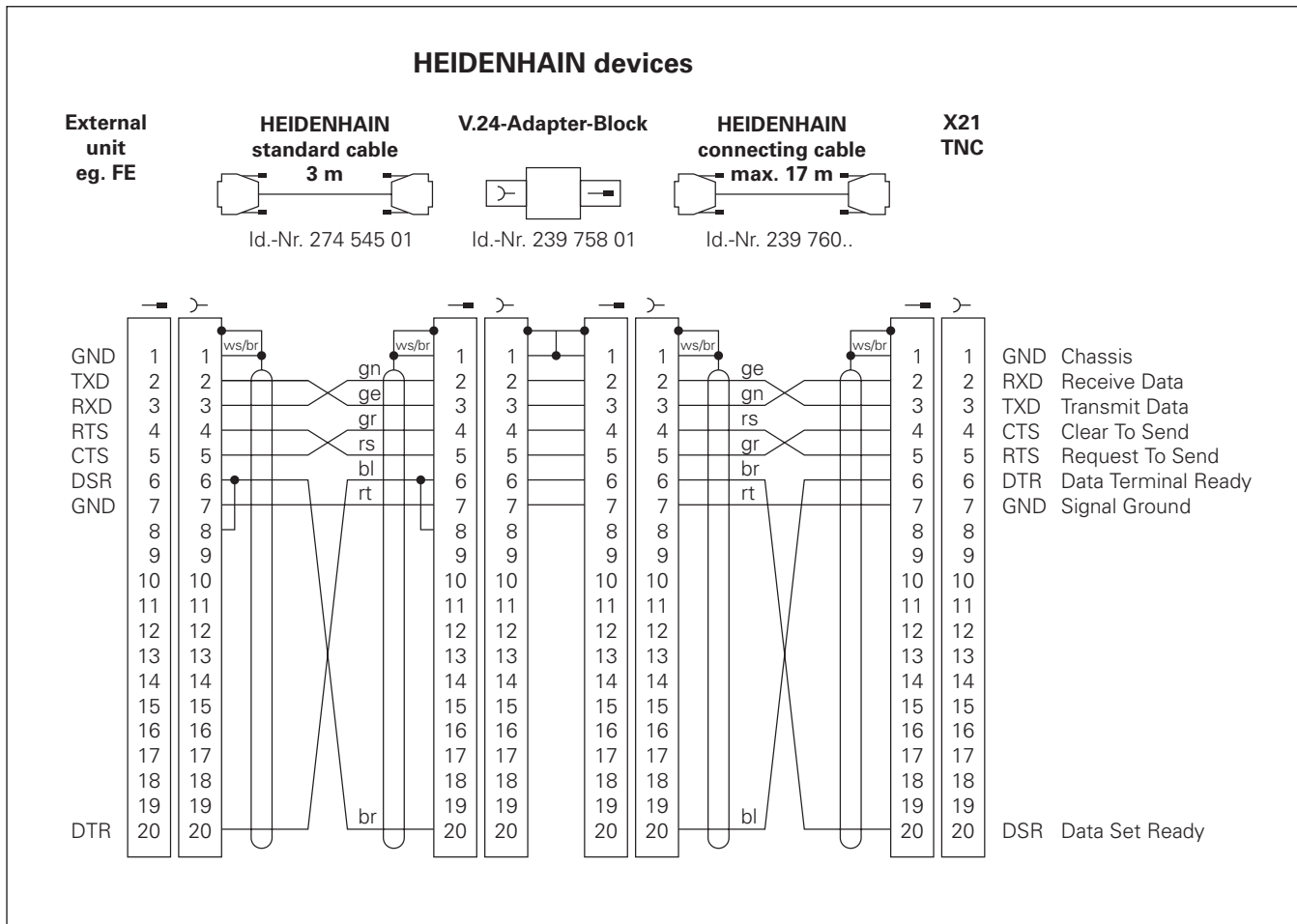
The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device, depending on the unit and type of data transfer.

The figure below shows the connector pin layout on the adapter block.

V.24-Adapter-Block
RS-232-C Adapter block

LE

Chassis GND	1	1	1	1	ws/br WH/BN			ws/br WH/BN	1	1	GND Chassis
TXD	2	2	2	2	ge	GN			2	2	RXD
RXD	3	3	3	3	gn	YL			3	3	TXD
RTS	4	4	4	4	rs	GY			4	4	CTS
CTS	5	5	5	5	gr	PK			5	5	RTS
DSR	6	6	6	6	br	BL			6	6	DTR
Signal GND	7	7	7	7	rt	RD			7	7	GND Signal
	8	8	8	8					8	8	
	9	9	9	9					9	9	
	10	10	10	10					10	10	
	11	11	11	11					11	11	
	12	12	12	12					12	12	
	13	13	13	13					13	13	
	14	14	14	14					14	14	
	15	15	15	15					15	15	
	16	16	16	16					16	16	
	17	17	17	17					17	17	
	18	18	18	18					18	18	
	19	19	19	19	bl	BN			19	19	
DTR	20	20	20	20					20	20	DSR

RS-422/V.11 Interface
Only non-HEIDENHAIN devices are connected to the RS-422 interface.

☞ The pin layouts on the TNC logic unit (X22) and on the adapter block are identical.

External unit e.g. PC	V.11-Adapter-Block	HEIDENHAIN-connecting cable max. 1000 m	X22 TNC

Id.-Nr. 249 819 01

Id.-Nr. 250 478..

1	1	1	BK BL BK	1	1 GND Chassis
2	2	2	GY	2	2 RXD
3	3	3	WH	3	3 CTS
4	4	4	GN	4	4 TXD
5	5	5	WH/GN	5	5 RTS
6	6	6	GY/PK	6	6 DSR
7	7	7	BK	7	7 DTR
8	8	8	RD	8	8 GND Signal
9	9	9	PK	9	9 RXD
10	10	10	BN	10	10 CTS
11	11	11	YL	11	11 TXD
12	12	12	BN/GN	12	12 RTS
13	13	13	RD/BL	13	13 DSR
14	14	14		14	14 DTR
15	15	15		15	15
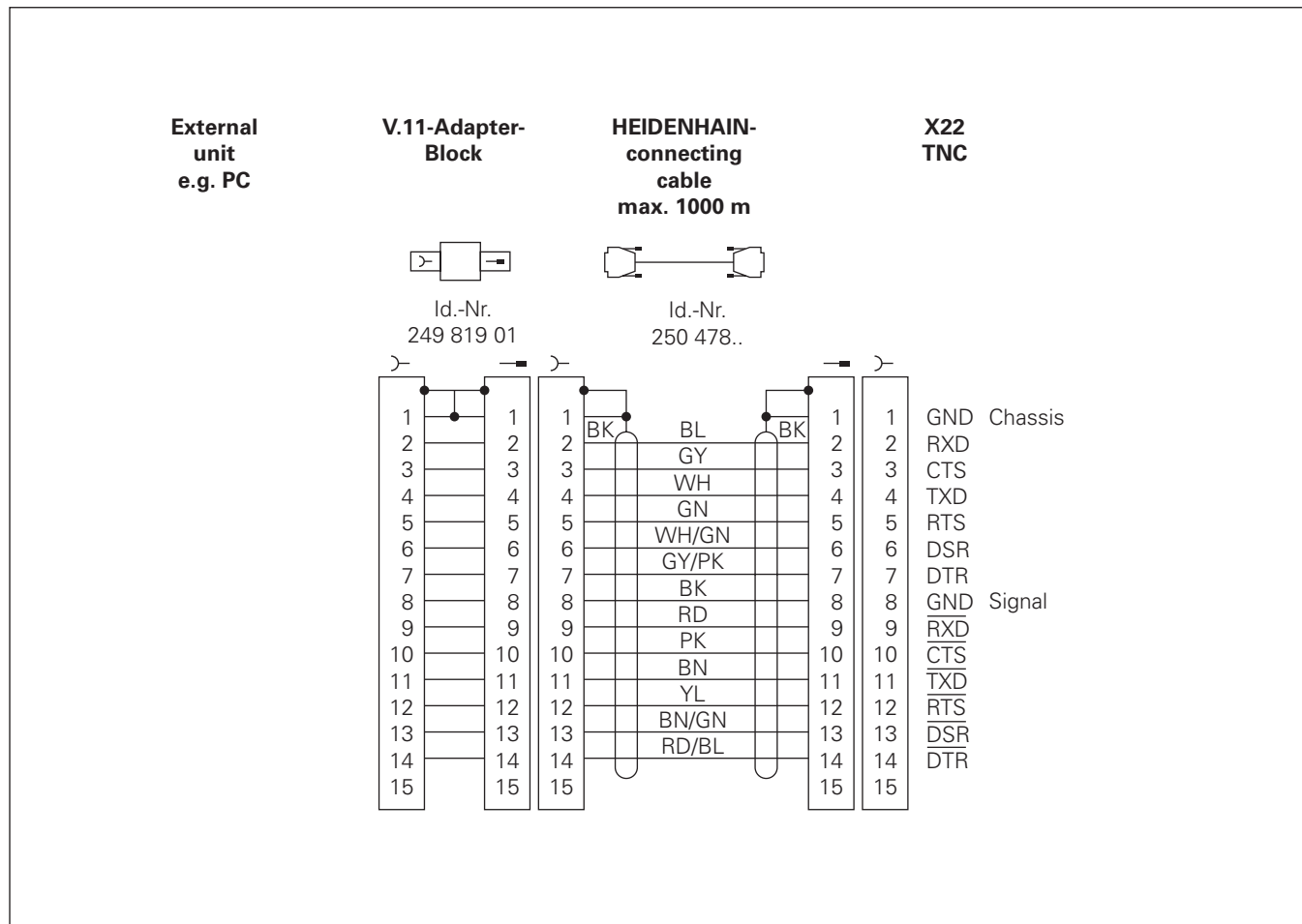
15.3 Technical Information

TNC features

Description	Contouring control for machines with up to 5 axes; with oriented spindle stop. The TNC 426 CA features analog speed control, the TNC 426 PA features digital speed control and integrated current controller
Components	■ Logic unit ■ Keyboard ■ Color visual display unit with soft keys
Data interfaces	■ RS-232-C / V.24 ■ RS-422 / V.11 Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with HEIDENHAIN software TNCREMO
Simultaneous axis control for contour elements	■ Straight lines: up to 5 axes Export versions TNC 426 CE, TNC 425 PE: 4 axes ■ Circles: up to 3 axes (with tilted working plane) ■ Helixes: 3 axes
Look Ahead	■ Defined rounding of discontinuous contour transitions (such as for 3-D surfaces) ■ Collision prevention with the SL cycle for open contours ■ Geometry precalculation of radius-compensated positions for feed rate adaptation
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Interactive programming graphics ■ Test run graphics ■ Program run graphics
File types	■ HEIDENHAIN conversational programming ■ ISO programming ■ Tool tables ■ Datum tables ■ Point tables ■ Pallet files ■ Text files ■ System files
Program memory	■ 170 MB hard disk for NC programs ■ No limit on number of files
Tool definitions	Up to 254 tools in the program or in tables
Programming support	■ Functions for approaching and departing the contour ■ On-screen pocket calculator ■ Structuring long programs

Programmable functions

Contour elements	■ Straight line ■ Chamfer ■ Circular arc ■ Circle center ■ Circle radius ■ Tangentially connecting arc ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure
FK free contour programming	For all contour elements not dimensioned for conventional NC programming
Three-dimensional tool radius compensation	For changing tool data without having to recalculate the program
Program jumps	■ Subprograms ■ Program section repeats ■ Main program as subprogram
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Milling and finishing rectangular and circular pockets ■ Cycles for milling linear and circular slots ■ Linear and circular hole patterns ■ Cycles for multipass-milling of flat and twisted surfaces ■ Milling pockets and islands from a list of subcontour elements ■ Cylindrical surface interpolation
Coordinate transformations	■ Datum shift ■ Mirroring ■ Rotation ■ Scaling factor ■ Tilting the working plane
3-D touch probe applications	■ Touch probe functions for setting datums and for automatic workpiece measurement ■ Digitizing 3-D surfaces with the measuring touch probe (optional) ■ Digitizing 3-D surfaces with the triggering touch probe (optional) ■ Automatic tool measurement with the TT 120
Mathematical functions	■ Basic arithmetic +, –, x and ÷ ■ Trigonometry sin, cos, tan, arcsin, arccos, arctan ■ Square root ($\sqrt{a}$) and root sum of squares ($\sqrt{a^2 + b^2}$) ■ Squaring (SQ) ■ Powers (^) ■ π (3.14) ■ Logarithms ■ Exponential function ■ Negation (NEG) ■ Forming an integer (INT) ■ Forming absolute numbers (ABS) ■ Dropping the values before the decimal point (FRAC) ■ Logical comparisons (greater than, less than, equal to, not equal to)

TNC error messages during test run and program run

ANGLE REFERENCE MISSING
- Complete your definition of the arc and its end points.
- If you enter polar coordinates, define the polar angle correctly.

ARITHMETICAL ERROR
You have calculated with non-permissible values.
- Define values within the range limits.
- Choose probe positions for the 3-D touch probe that are farther apart.
- Enter a value other than 0 for the number of teeth in the tool table when you are measuring the individual teeth with the TT touch probe.
- Calibrate TCH PROBE 30 (calibrate TT) before measuring the tool length or radius.
- All calculations must be mathematically possible.

AXIS DOUBLE PROGRAMMED
Each axis can have only one value for position coordinates.

BLK FORM DEFINITION INCORRECT
- Program the MIN and MAX points according to the instructions.
- Choose a ratio of sides that is less than 200:1.

CALIBRATE TOUCH PROBE
- Recalibrate the TT, machine parameters for the TT have been changed.
- Changed machine parameters for measuring touch probe: recalibrate the measuring touch probe

CHAMFER NOT PERMITTED
A chamfer block must be located between two straight-line blocks with identical radius compensation.

CIRCLE CENTER UNDEFINED
- Define a circle center with the CC key.
- Define a pole with the CC key.

CIRCLE END POS. INCORRECT
- Enter complete information for connecting arc.
- Enter end points that lie on the circular path.

CYCL INCOMPLETE
- Define the cycles with all data in the proper sequence.
- Do not call the coordinate transformation cycles.
- Define a cycle before calling it.
- Enter a pecking depth other than 0.

EXCESSIVE SUBPROGRAMMING
- Conclude subprograms with LBL0.
- Program CALL LBL for subprograms without REP.
- Program CALL LBL for program section repeats to include the repetitions (REP).
- Subprograms cannot call themselves.
- Subprograms cannot be nested in more than eight levels.
- Main programs cannot be nested as subprograms in more than four levels.

FAULTY PROGRAM DATA
The program that was read in through the data interface contains incorrect block formats.

FEED RATE IS MISSING
- Enter the feed rate for the positioning block.
- Enter FMAX in each block.

GROSS POSITIONING ERROR
The TNC monitors positions and movements. If the actual position deviates excessively from the nominal position, this blinking error message is displayed. You must press the END key for a few seconds to acknowledge the error message (system reset).

KEY NON-FUNCTIONAL
This message always appears when you press a key that is not needed for the current dialog.

LABEL NUMBER NOT FOUND	Only call label numbers that have been set.
NO EDITING OF RUNNING PROGRAM	A program cannot be edited while it is being transmitted or executed.
PATH OFFSET WRONGLY ENDED	Do not cancel tool radius compensation in a block with a circular path.
PATH OFFSET WRONGLY STARTED	■ Use the same radius compensation before and after an RND and CHF block. ■ Do not begin tool radius compensation in a block with a circular path.
PGM-SECTION CANNOT BE SHOWN	■ Enter a smaller tool radius. ■ 4-D and 5-D movements cannot be graphically simulated. ■ Enter a tool axis for simulation that is the same as the axis in the BLK FORM.
PLANE WRONGLY DEFINED	■ Do not change the tool axis while a basic rotation is active. ■ Correctly define the main axes for circular arcs. ■ Define both main axes for CC.
PROBE SYSTEM NOT READY	■ Be sure the transmitting/receiving window of the TS 630 is oriented to the receiving unit. ■ Check whether the touch probe is ready for operation.
PROGRAM START UNDEFINED	■ Begin the program only with a TOOL DEF block. ■ Do not resume an interrupted program at a block with a tangential arc or if a previously defined pole is needed.
RADIUS COMP. UNDEFINED	Enter radius compensation RR or RL in the first subprogram for Cycle 14 CONTOUR GEOMETRY.
ROUNDING-OFF NOT PERMITTED	Enter tangentially connecting arcs and rounding arcs correctly.
ROUNDING RADIUS TOOL LARGE	Rounding arcs must fit between contour elements.
SCALING FACTOR NOT PERMITTED	Enter identical scaling factors for coordinate axes in the plane of the circle.
SELECTED BLOCK NOT ADDRESSED	Before a test run or program run, you must enter GOTO 0.
STYLUS ALREADY IN CONTACT	Before probing, pre-position the stylus where it is not touching the workpiece surface.
TOOL LIFE EXPIRED	TIME1 or TIME2 of TOOL.T has been exceeded with no replacement tool defined in the tool table.
TOOL RADIUS TOO LARGE	Enter a tool radius that ■ lies within the given limits ■ permits the contour elements to be calculated and machined.
TOUCH POINT INACCESSIBLE	■ Pre-position the 3-D touch probe to a position nearer the model. ■ The machine parameters that define the position of the TT touch probe do not correspond to the actual position of the TT touch probe.
WRONG AXIS PROGRAMMED	■ Do not attempt to program locked axes. ■ Program a rectangular pocket or slot in the working plane. ■ Do not mirror rotary axes. ■ Enter a positive chamfer length.
WRONG RPM	Program a spindle speed within the programmed range.
WRONG SIGN PROGRAMMED	Enter the correct sign for the cycle parameter.

TNC error messages during digitizing

AXIS DOUBLE PROGRAMMED	Program two different axes for the coordinates of the starting point (CONTOUR LINES cycle).
EXCHANGE TOUCH PROBE BATTERY	■ Change the battery in the probe head (TS 630). ■ This message is output when the touch probe reaches the end of a line.
FAULTY RANGE DATA	■ Enter MIN coordinates that are smaller than their MAX coordinates. ■ Define the RANGE within the limits through software limit switches. ■ Define the range for the MEANDER and CONTOUR LINES cycles.
INCORRECT ENTRY IN MP6322	Check the values defined in machine parameters 6322.0 to 6322.2.
MIRRORING NOT PERMITTED	Reset all coordinate transformations before digitizing.
PLANE WRONGLY DEFINED	Define the starting position coordinate (CONTOUR LINES cycle) in axes different from the stylus axis.
PROBE SYSTEM NOT READY	■ Be sure the transmitting/receiving window of the TS 630 is oriented to the receiving unit. ■ Check whether the touch probe is ready for operation. ■ Touch probe cannot be retracted. ■ With measuring touch probes: One or several axes of the measuring touch probe are defective. Contact your repair service agency.
RANGE EXCEEDED	Enter a RANGE that includes the entire 3-D surface to be scanned.
ROTATION NOT PERMITTED	Reset all coordinate transformations before digitizing.
SCALING FACTOR NOT PERMITTED	Reset all coordinate transformations before digitizing.
START POSITION INCORRECT	Program the starting point coordinates for the CONTOUR LINES cycle within the RANGE.
STYLUS ALREADY IN CONTACT	Pre-position the touch probe so that the stylus cannot be deflected outside the RANGE.
TIME LIMIT EXCEEDED	Enter a TIME LIMIT that is appropriate to the 3-D surface to be scanned (CONTOUR LINES cycle).
TOO MANY POINTS	You can define up to 893 points in the PNT file; scan the digitizing range again and, if necessary, program a larger probe point interval.
TOUCH POINT INACCESSIBLE	■ The stylus must not be deflected before it reaches the RANGE. ■ The stylus must be deflected somewhere within the RANGE.
WRONG AXIS PROGRAMMED	■ Enter calibrated touch probe axis in the RANGE cycle. ■ Enter the correct rotary axis in the RANGE cycle. ■ Do not program the same axes twice in the RANGE cycle.

Index

M	Effect of M Function	Effective at block start	end	Page
M00	Stop program run/Spindle STOP/Coolant OFF		■	**117**
M02	Stop program run/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	**117**
M03	Spindle ON clockwise	■		
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	**117**
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	**117**
M08	Coolant ON	■		
M09	Coolant OFF		■	**117**
M13	Spindle ON clockwise/Coolant ON	■		
M14	Spindle ON counterclockwise/Coolant ON	■		**117**
M30	Same function as M02		■	**117**
M89	Vacant miscellaneous function **or** Cycle call, modally effective (depending on machine parameter)	■	■	**205**
M90	Only in lag mode: Constant contouring speed at corners		■	**119**
M91	Within the positioning block: Coordinates are referenced to machine datum		■	**117**
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		**117**
M94	Reduce display of rotary axis to value less than 360°	■		**126**
M97	Machine small contour steps		■	**122**
M98	Completely machine open contours		■	**123**
M99	Blockwise cycle call		■	**205**
M101	Automatic tool change with replacement tool if maximum tool life has expired	■		
M102	Reset M101		■	**61**
M103	Reduce feed rate during plunging to factor F (percentage)	■		**123**
M105	Machining with first kv factor	■		
M106	Machining with second kv factor	■		**311**
M107	Suppress error message for replacement tool with oversize	■		
M108	Reset M107		■	**61**
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)	■		
M110	Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only)	■		
M111	Reset M109/M110		■	**124**
M112	Automatic insertion of rounding arcs at non-tangential straight-line transitions; Enter tolerance T for contour deviation	■		
M113	Reset M112		■	**120**
M114	Automatic compensation of machine geometry during operation with tilting axes	■		
M115	Reset M114		■	**127**
M116	Feed rate for angular axes in mm/min	■		**125**
M118	Superimpose handwheel positioning during program run	■		**125**
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		**124**
M124	Ignore points for calculating rounding arc with M112	■		**121**
M126	Shorter-path traverse of rotary axes	■		
M127	Reset M126		■	**126**
M132	Jolt reduction when changing the direction of traverse	■		
M133	Reset M132		■	**121**
M200	Laser cutting: Output programmed voltage directly	■		
M201	Laser cutting: Output voltage as function of distance	■		
M202	Laser cutting: Output voltage as function of speed	■		
M203	Laser cutting: Output voltage as function of time (ramp)	■		
M204	Laser cutting: Output voltage as function of time (pulse)	■		**128**

HEIDENHAIN